Module 1: Basics of Solids Modeling with SolidWorks

Introduction

SolidWorks is the state of the art in computer-aided design (CAD). SolidWorks represents an object in a virtual environment just as it exists in reality, i.e., having volume as well as surfaces and edges. This, along with exceptional ease of use, makes SolidWorks a powerful design tool. Complex three-dimensional parts with contoured surfaces and detailed features can be modeled quickly and easily with Solid-Works. Then, many parts can be assembled in a virtual environment to create a computer model of the finished product. In addition, traditional engineering drawings can be easily extracted from the solids models of both the parts and the final assembly. This approach opens the door to innovative design concepts, speeds product development, and minimizes design errors. The result is the ability to bring high-quality products to market very quickly.

CONSTRAINT-BASED SOLIDS MODELING

The *constraint-based solids modeling* used in SolidWorks makes the modeling process intuitive. The 3-D modeling begins with the creation of a 2-D **sketch** of the profile for the cross section of the part. The sketch of the cross section begins much like the freehand sketch of the face of an object. The initial sketch need not be particularly accurate; it needs only to reflect the basic geometry of the part's cross-sectional shape. Details of the cross section are added later. The next step is to **constrain** the two-dimensional sketch by adding *enough dimensions* and *parameters* to completely define the shape and size of the two-dimensional profile. The name *constraint-based modeling* arises because the shape of the initial two-dimensional sketch is "constrained" by adding dimensions to the sketch. Finally, a three-dimensional object is created by *revolving* or *extruding* the two-dimensional sketched profile. Figure 1 shows the result of revolving a simple L-shaped cross section by 270^{o} about an axis and extruding the same L-shaped cross section along an axis.

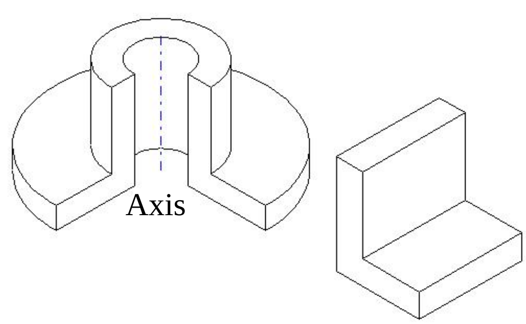


Figure 1 Revolved and extruded solid models of an L-shaped cross-section.

In either case, these solid bodies form the basic geometric solid shapes of the part. Other features can be added subsequently to modify the basic solid shape. Once the solids model is generated using SolidWorks, all of the surfaces have been automatically defined, so it is possible to shade it in order to create a photorealistic appearance. It is also easy to generate two-dimensional orthographic views of the object. Solid modeling is like the sculpting of a virtual solid volume of material. Because the volume of the object is properly represented in a solids model, it is possible to slice through the object and show a view of the object that displays the interior detail (sectional views). Once several solid objects have been created, they can be assembled in a virtual environment to confirm their fit and to visualize the assembled product. Solids models are useful for purposes other than visualization. The solids model contains a complete mathematical representation of the object, inside and out. This mathematical representation is easily converted into specialized computer code that can be used for stress analysis, heat-transfer analysis, fluid-flow analysis, and computer-aided manufacturing.

Getting Started in SolidWorks

Introduction and Reference

SolidWorks Corporation developed SolidWorks® as a three-dimensional, feature-based, solids-modeling system for personal computers. Solid modeling represents objects in a computer as volumes, rather than just as collections of edges and surfaces. Features are three-dimensional geometries with direct analogies to shapes that can be machined or manufactured, such as holes or rounds. Feature-based solid modeling creates and modifies the geometric shapes of an object in a way that represents common manufacturing processes. This makes SolidWorks a very powerful and effective tool for engineering design.

As with other computer programs SolidWorks organizes and stores data in files. Each file has a name followed by a period (dot) and an extension. There are several file types used in SolidWorks, but the most common file types and their extensions are

Part files	.prt or .sldprt
Assembly files	.asm or .sldasm
Drawing files	.drw or .slddrw

Part files are the files of the individual parts that are modeled. Part files contain all of the pertinent information about the part. Because SolidWorks is a solids-modeling program, the virtual part on the screen will look very similar to the actual manufacture part.

Assembly files are created from several individual part files that are virtually assembled (in the computer) to create the finished product.

Drawing files are the two dimensional engineering drawing representations of both the part and assembly file. The drawings should contain all of the necessary information for the manufacture of the part, including dimensions, part tolerances, and so on.

The part file is the ***driving*** file for all other file types. The modeling procedure begins with part files. Subsequent assemblies and drawings are based on the original part files. One advantage of SolidWorks files is the feature of dynamic links. Any change to a part file will automatically be updated in any corresponding assembly or drawing file.

Therefore, both drawing and assembly files must be able to find and access their corresponding part files in order to be opened. SolidWorks uses information embedded within the file and the filename to maintain these links automatically.

Starting SolidWorks

SolidWorks runs on computers running the Microsoft Windows® operating system. You open SolidWorks in the same way that you would start any other program.

Checking the Options Settings

The SolidWorks window that appears on the computer screen looks similar to the standard Microsoft Windows interface, as shown in Figure 2. The top line of the window is the Menu bar from which menus for various operations can be opened. Below the Menu bar are the toolbars which provide access to a variety of commonly used operations, or tools, with a single click of the mouse button. Toolbars can also extend down the right and left sides of the window. They may or may not be shown on your screen. At this point, most items within the toolbars are grayed out. This indicates that they are not presently available for use.

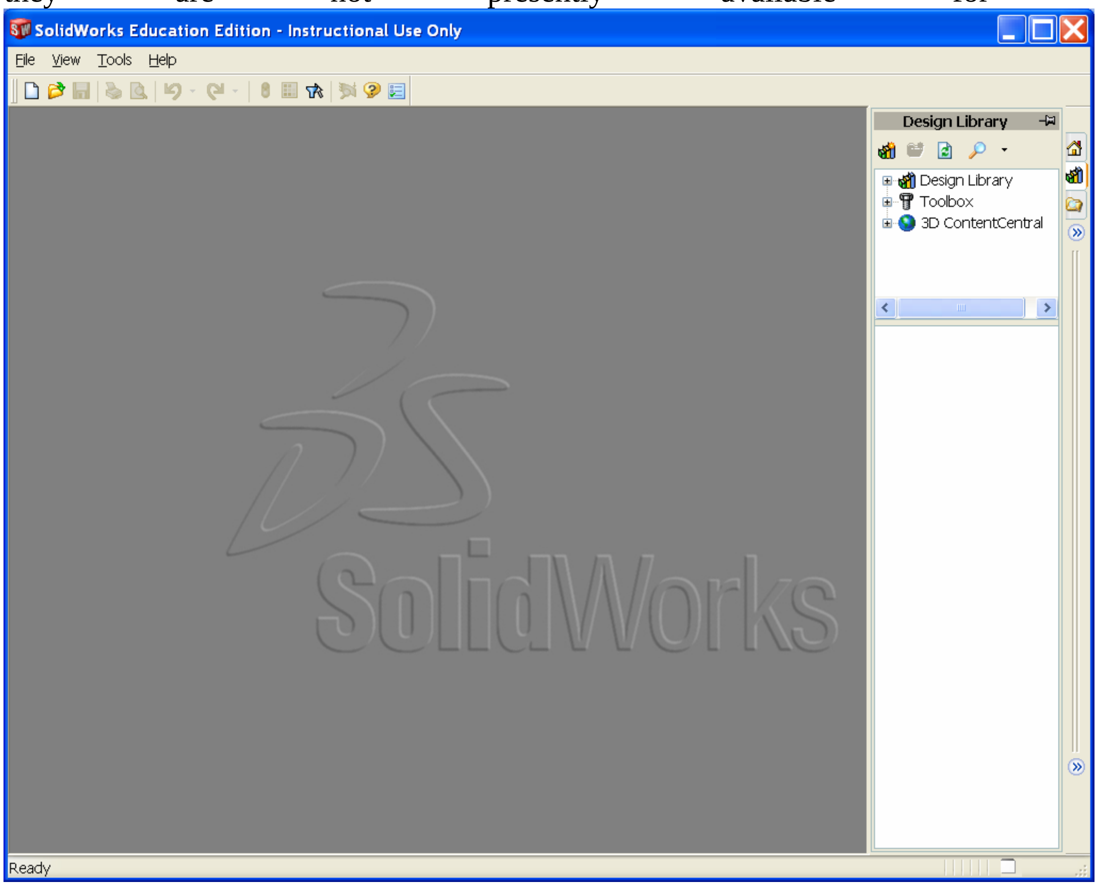


Figure 2 SolidWorks window.

Tool bars

To specify which toolbars are displayed on the screen, select View **Tool-bars** (i.e., select **Toolbars** from the **View** menu). Be sure that the **Features, Sketch, Standard, Standard Views** and **View** toolbars are checked, as shown in Figure 3. If they are not, click on each of these items until all are checked. If other toolbars are checked, click on them to uncheck them. It may be necessary to select **View,** then **Toolbars** again to display the menu after checking (or un-checking) an item to confirm that the desired change was made. The **Standard Views toolbar** may appear as a dialog box in the Graphics Window instead of as a toolbar. If so, click the blue bar at the top of the dialog box with the left mouse button and drag it to the toolbar at the upper right of the Graphics Window. Release the mouse button. The dialog box should change to a toolbar.

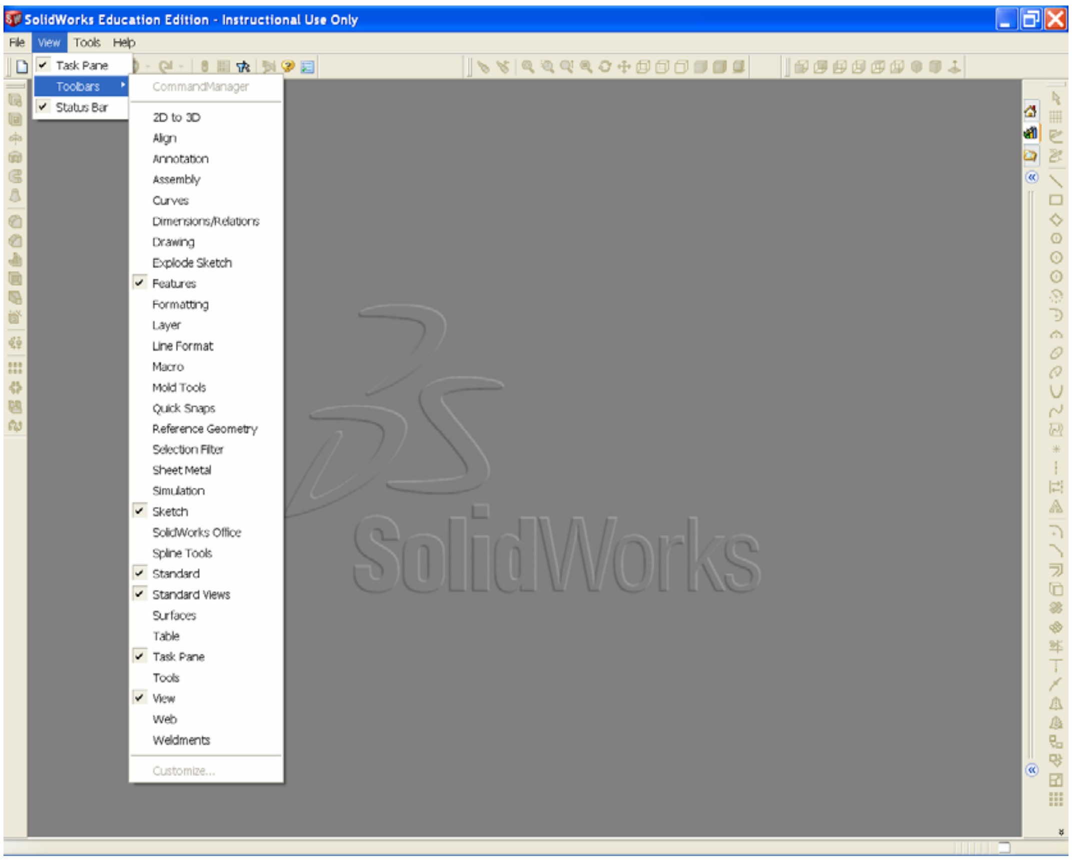


Figure 3 Toolbars menu.

- The **Sketch** toolbar contains tools to draw lines, circles, rectangles, arcs, and so on.
- The **Features** toolbar contains tools that modify sketches and existing features of a part.
- The **Standard** toolbar contains the usual commands available for manipulating files (Open, Save, Print, and so on), editing documents (Cut, Copy, and Paste), and accessing Help.

- The **Standard Views** toolbar contains common orientations for a model.
- The **View** toolbar contains tools to orient and rescale the view of a part.

You can find these toolbars around the Graphics Window by checking and un-checking them in the **View**⇒**Toolbars** menu. The toolbars will appear as you check them and disappear as you uncheck them. Currently, most of the items in the tool-bars are grayed out, since they are unusable. They will become active when they are available for use. Be sure that the **Toolbars** menu looks like the one in Figure 3 before continuing. Click any open spot in the Graphics Window to close all menus.

Getting Help

If you have questions while you are using SolidWorks, you can find answers in several ways:

- Click **SolidWorks Help Topics** in the **Help** Menu bar.
- Move the cursor over a toolbar button to see the ToolTip, which indicates the name of the tool.
- Move the cursor over buttons or click menu items. The Status bar at the bottom of the SolidWorks window will provide a brief description of the function.
- Click the ***Help*** button in a dialog box.
- Refer to the *Solid Works User's Guide,* by SolidWorks Corporation, for detailed information.

Creating a New Part

With the SolidWorks window open, select **File**⇒**New** in the Menu bar, or click the New button (a blank-sheet icon) in the **Standard** toolbar.

The ***New Solid Works Document*** dialog box appears as shown in Figure 4. You will be modeling a new part. If *Part* is already highlighted, click ***OK.*** If **it is not** highlighted, click *Part,* then ***OK.***

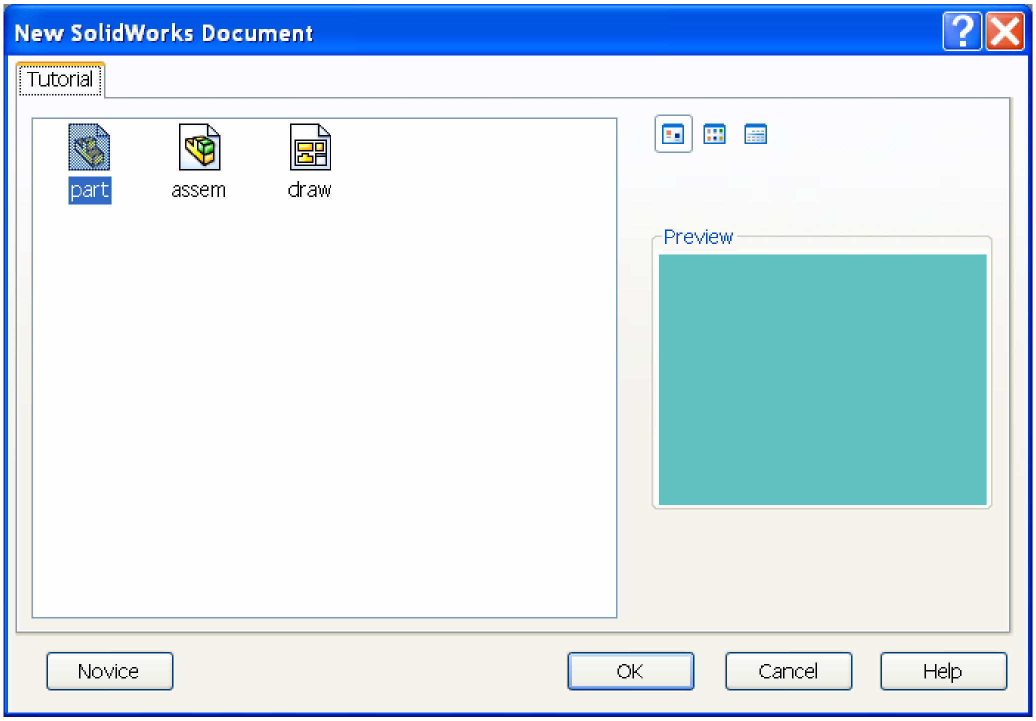


Figure 4 New document window.

A new window appears with the name Partl, as shown in Figure 5. On the left side of the window is the Feature Manager Design tree.

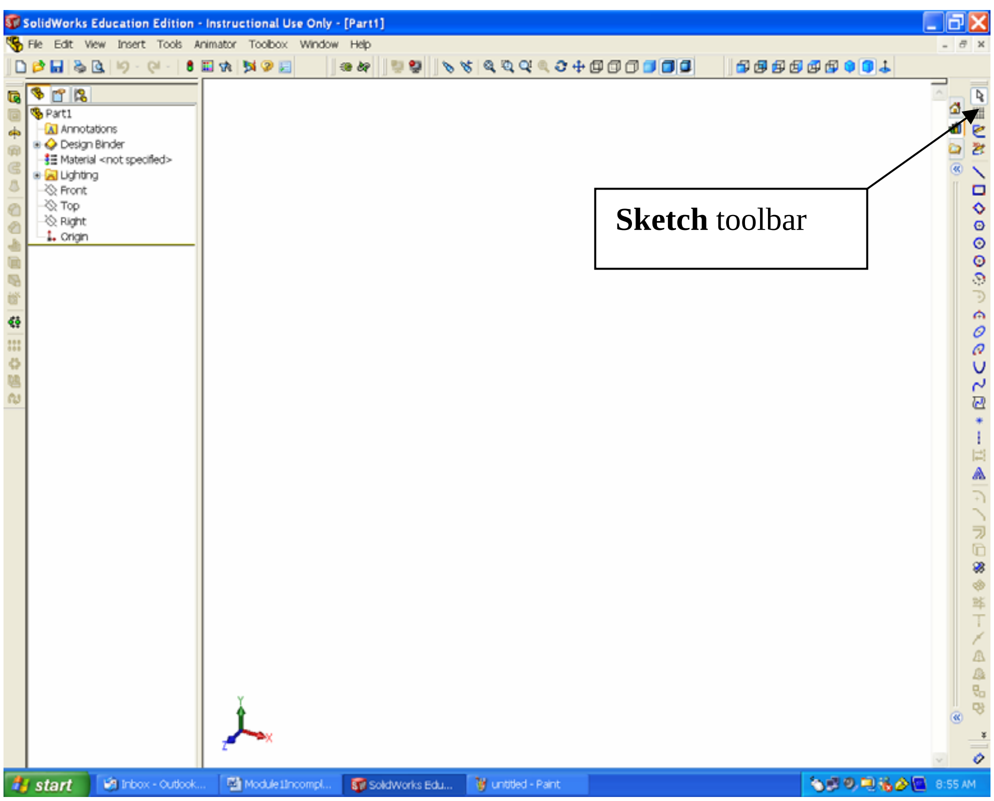


Figure 6 New part window.

It contains a list of the features that have been created so far. Every new part starts with six features: annotations, lighting, three datum planes, and an origin. The datum planes are three mutually perpendicular planes that are created in space as references for constructing features of the part that you are modeling. The three planes intersect at the origin, which is in the center of the Graphics Window. The arrows in the lower left corner of the Graphics Window show the coordinate directions. As the part is modeled, the features that are created will appear in the Feature Manager Design tree. These features can be highlighted or modified by clicking on them in the Feature Manager Design tree. For example, click on a plane or the origin in the Feature Manage Design tree to highlight these items. **Front** is the plane of the screen, **Top** is the horizontal plane perpendicular to the screen, and **Right** is the vertical plane perpendicular to the screen. Finish by clicking on Part1 in the Feature Manager Design tree, so that no plane is highlighted.

Sketching

Every part begins as a cross section sketched on a two-dimensional plane. Once a sketch is made, it is extruded or revolved into the third dimension to create a three-dimensional object. This is the base feature of the part.

The **Sketch** toolbar, shown in Figure 6, has tools to set up and manipulate a sketch of a cross section. Find the **Sketch** toolbar. Move the cursor over each of the tools, but do not click on any of the tools. The Tool Tips should appear, displaying the name of each tool.

- **Select** highlights sketch entities; drags sketch entities and endpoints and modifies dimension values.
- **Grid** activates the ***Grid/Snap*** field of the ***Document Properties*** dialogue box to change the sketching environment.
- **Dimension** adds dimensions to sketch entities.
- **Sketch** opens and closes sketches as a part is created.

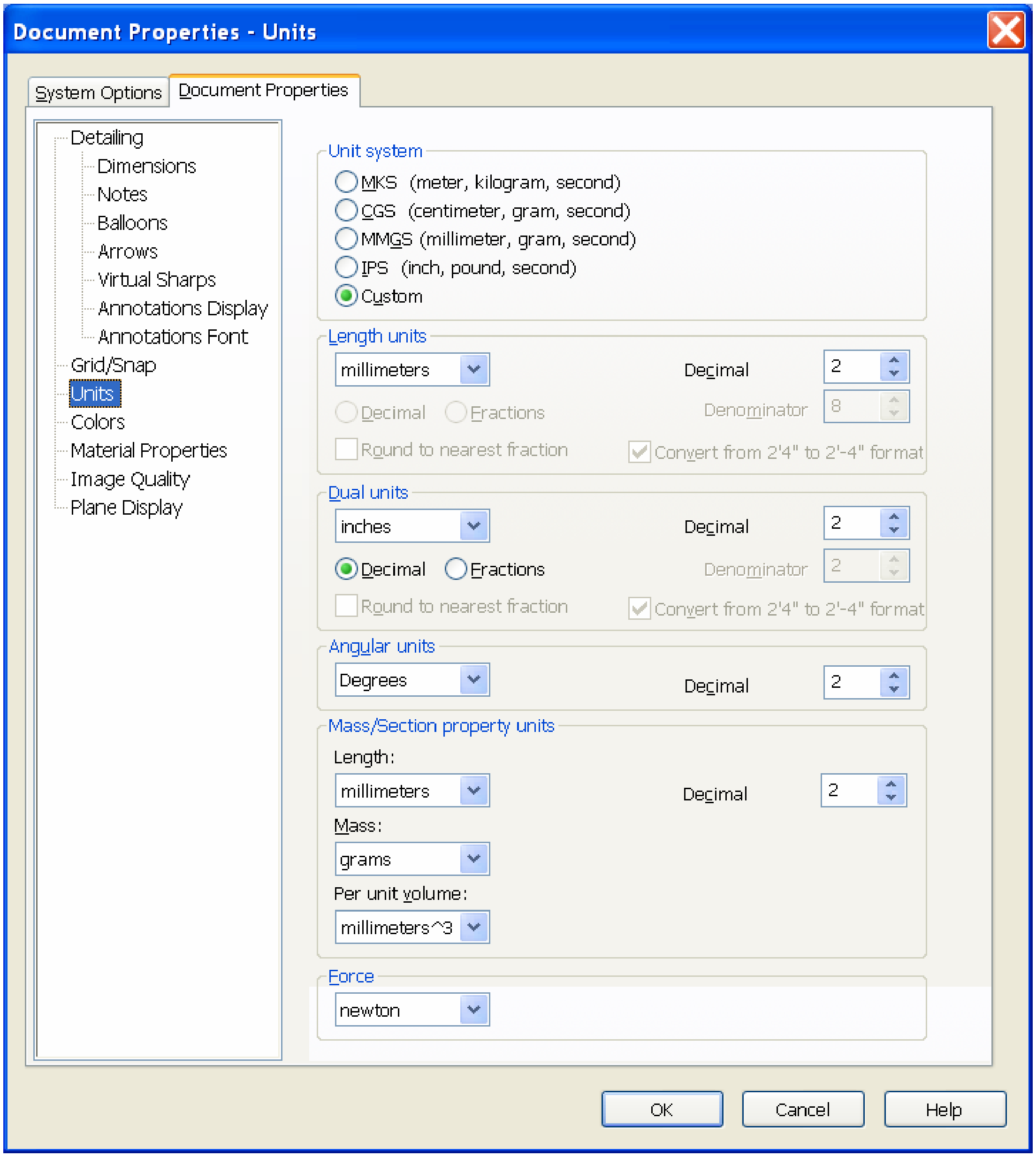


Figure 6 Document properties dialog box: ***Units***

To set the units and grid size to be used, click the **<u>Grid</u>** toolbar button with the left mouse button. Document properties dialog box shown in Figure 6 will appear. Click ***Units*** on the left side of the dialog box to set the units. Setup appropriate units (*inches* or *mm*) with the desired number of decimal places or fraction denominator.

Click ***Grid/Snap*** on the left side of the dialogue box to control the grid that will appear on the screen when a cross-section is sketched. Be sure that all three of the boxes under *Grid* are checked, as shown in Figure 7. Adjust the grid spacing to desired values.

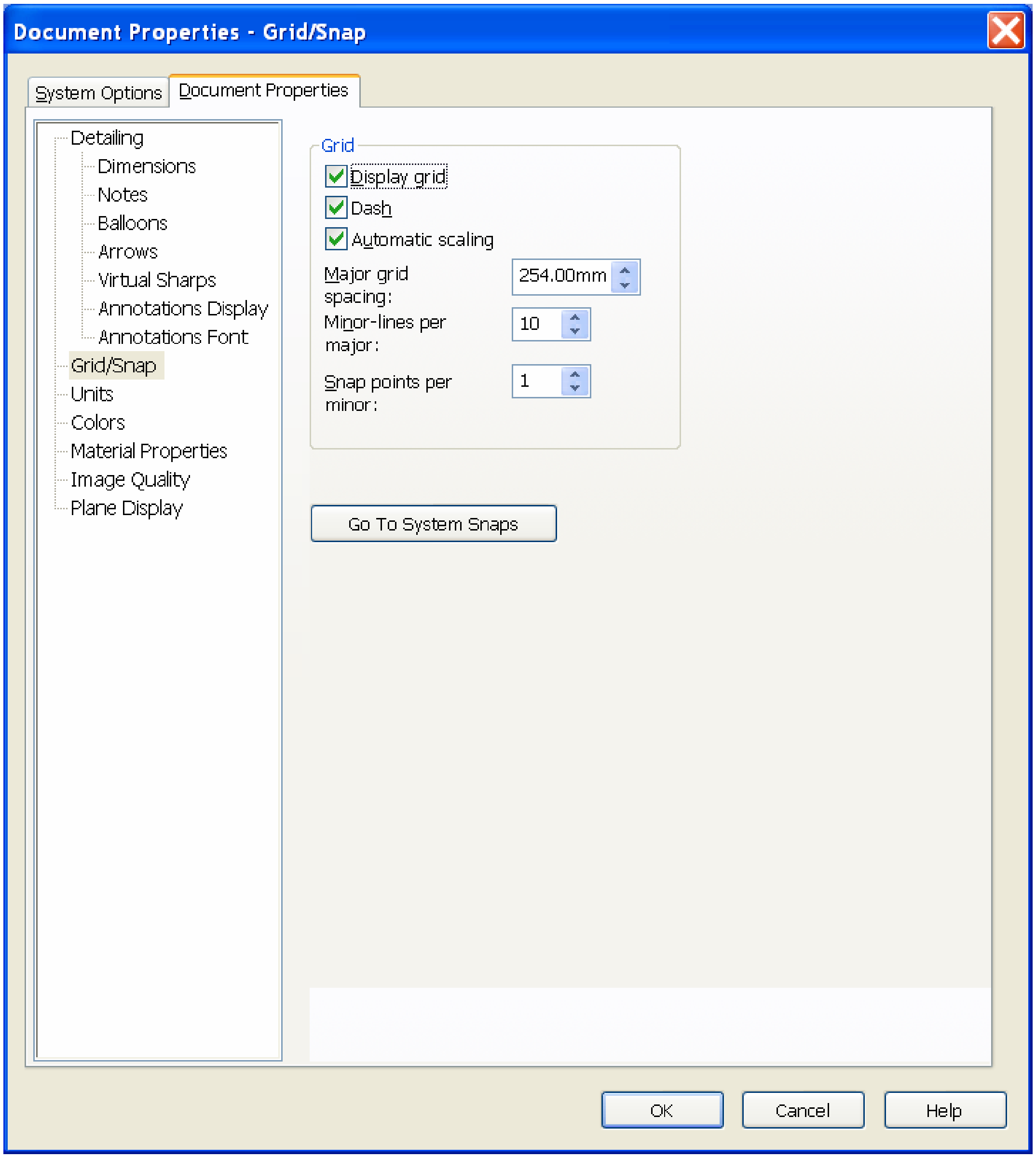


Figure 7 Document properties dialog box: ***Grid/Snap***

Snap controls the way in which sketched lines are related to the grid. The points that are sketched should "snap" to the nearest intersection of grid lines when they are close. Click **Go To System Snaps**. If not, click the box **Grid** to check it as shown in Figure 8.

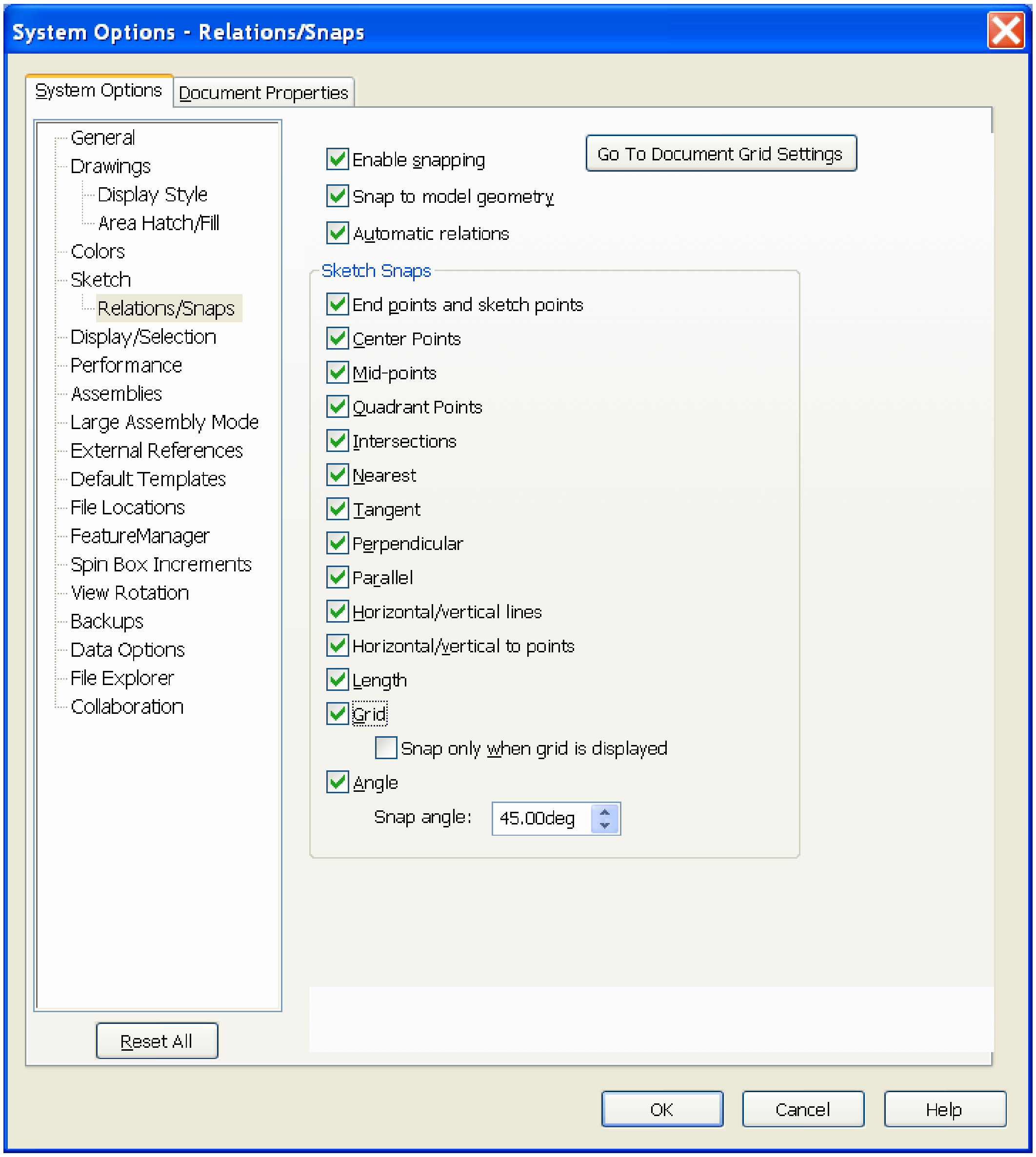


Figure 8 Document properties dialog box: **System Snap**

Click ***Detailing*** on the left side of tile dialogue box. If necessary, change ***Dimensioning standard*** to *ANSI* for inch units and ISO in case of mm units. Then, click ***OK*** at the bottom of the dialogue box to accept the values.

Open a new sketch by selecting **Insert Sketch,** or by clicking the **<u>Sketch</u> button (a pencil** drawing a line) in the **<u>Sketch</u>** toolbar. (Note that, for most commands in SolidWorks, it is possible to initiate the command from either the Menu bar or the toolbars.) A grid should appear on the screen, as shown in Figure 9, indicating that the sketch mode is active. The window's name changes to **Sketch of Part1.** In the bottom right corner of the screen, the Status bar reads *Editing Sketch.* You are now ready to sketch in the **Front** plane.

The **<u>Sketch</u>** toolbar, shown in Figure 9, contains tools to create and modify two-dimensional features, called Sketch Entities. Sketch Entities are items that can be drawn on the sketch. The following Sketch Entities and Sketch Tools are available:

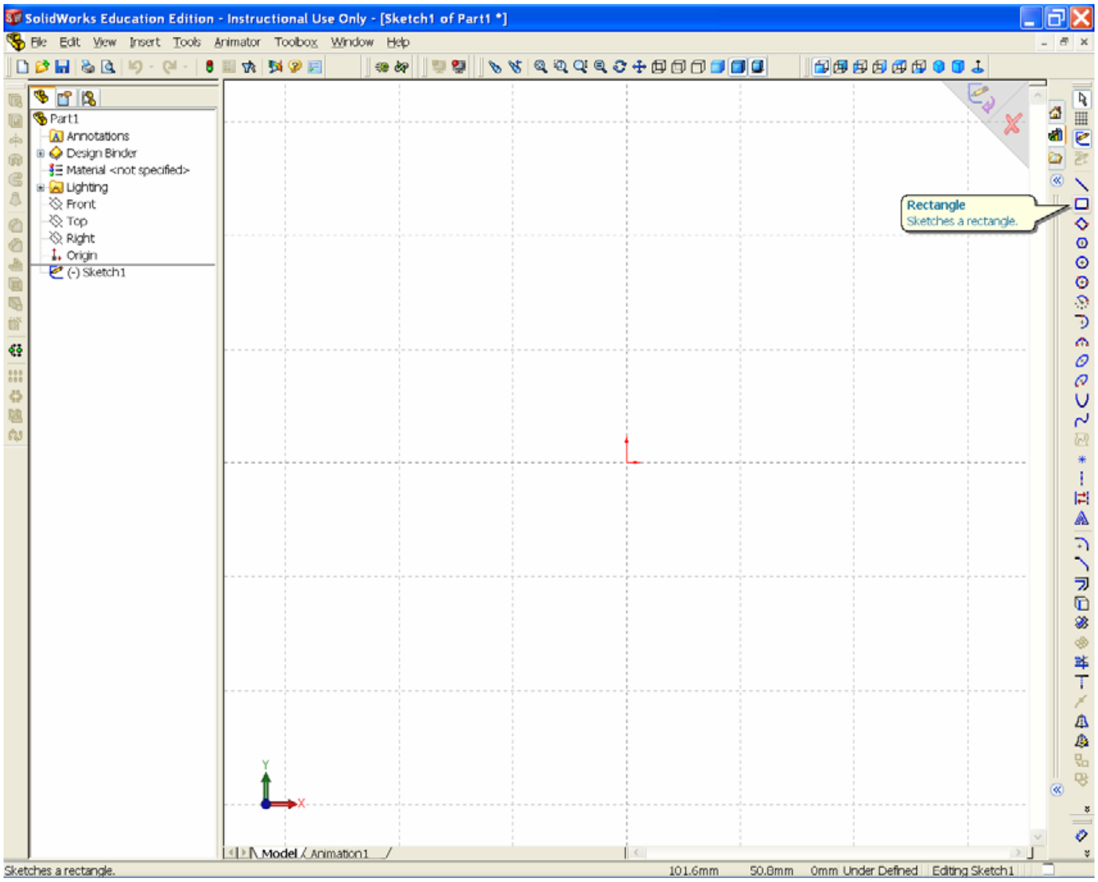
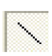
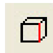


Figure 9 New sketch window.

- **Line**  creates a straight line.
- **Centerpoint Arc**  creates a circular arc from a center point, a start point, and an end point.
- **Tangent Arc**  creates a circular arc tangent to an existing sketch entity.
- **3 Pt Arc**  creates a circular arc through three points.
- **Circle**  creates a circle.
- **Spline** creates a curved line that is not a circular arc.
- **Polygon creates** a regular polygon.
- **Rectangle**  creates a rectangle.
- **Point**  creates a reference point that is used for constructing other sketch entities.

- **<u>Centerline</u>** creates a reference line that is used for constructing other sketch entities.
- **<u>Convert Entities</u>**  creates a sketch entity by projecting an edge, curve, or contour onto the sketch plane.
- **<u>Mirror</u>**  reflects entities about a centerline.
- **<u>Fillet</u>**  creates a tangent arc between two sketch entities by rounding an inside or an outside corner.
- **<u>Offset Entities</u>**  creates a sketch curve that is offset from a selected sketch entity by a specified distance.
- **<u>Trim</u>**  removes a portion of a line or curve.
- **<u>Construction Geometry</u>** creates entities that aid in sketching.
- **<u>Linear Sketch Step and Repeat</u>** creates a linear pattern of sketch entities.
- **<u>Circular Sketch Step and Repeat</u>** creates a circular pattern of sketch entities.

Move and hold the cursor over each of the tools to display its function but do not click on the tool. Note the description of each tool in the Status bar at the bottom of the SolidWorks window. Some of these tools may not he included in the toolbar or other tools may be available, depending on the way in which it was previously set up. All tools are available in the **Tools** Menu.

Module 2a: Basic Sketching I

In this tutorial, you will draw the sketch of the model shown in Figure 1. The sketch is shown in Figure 2. You will not dimension the sketch. The solid model and the dimensions are given only for your reference.

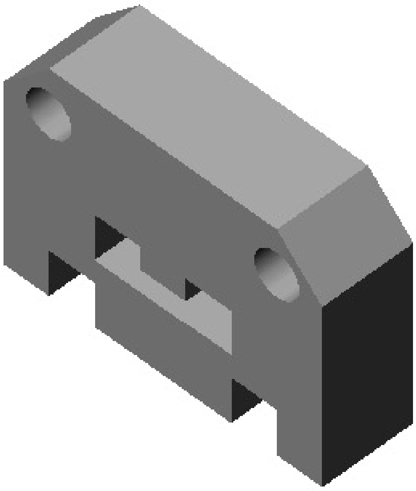


Figure 1 Solid model

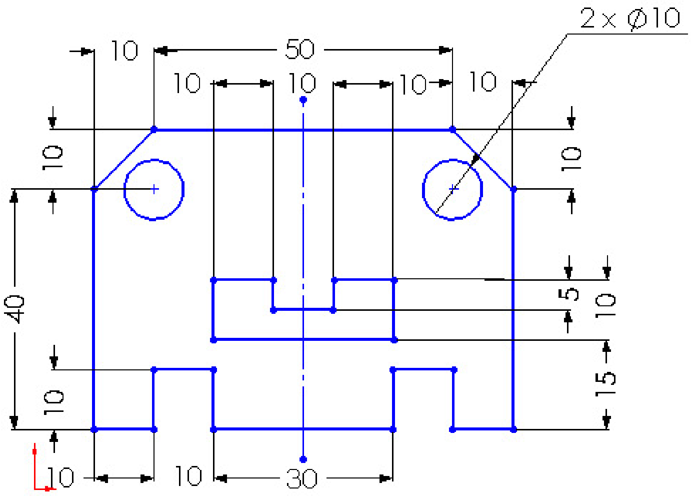


Figure 2 Sketch for the solid model

The steps that will be followed to complete this tutorial are listed below:

1. Start SolidWorks and then start a new part document.
2. Maximize the part file document and then switch to the sketching environment.
3. Draw the sketch of the model using the Line and the Circle tools, refer to Figures 4 through 7.
4. Save the sketch and then close the document.

Starting SolidWorks and Starting a New Part Document

1. Start SolidWorks by choosing **Start> Programs> SolidWorks 2005 > SolidWorks 2005** or by double-clicking the shortcut icon  of SolidWorks available on the desktop of your computer.
2. The SolidWorks 2005 window is displayed. Choose the New Document option from this window. The New SolidWorks Document dialog box is displayed.
3. The Part button  is chosen by default. Choose the OK button from the New SolidWorks Document dialog box as shown in Figure 3. A new SolidWorks part document will be opened. But the part document window will not be maximized in the SolidWorks window.

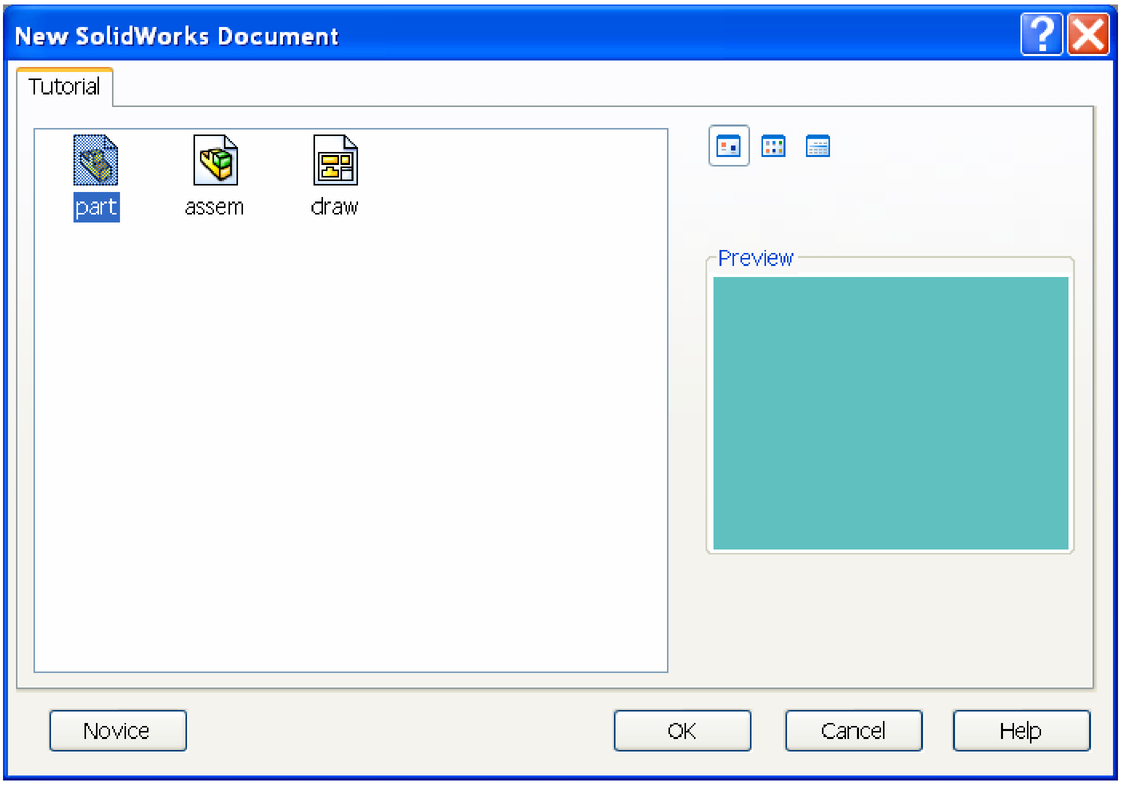


Figure 3 New SolidWorks document dialog box.

4. Choose the Maximize  button available on the upper right corner of the part document window to maximize the document window. When you open a new part document, the part modeling environment is active by default. As you first need to draw the sketch of the feature, you need to invoke the sketching environment.

5. Select the **Front** plane.

6. Choose the **Sketch**  button from the Standard toolbar. The **Sketch tools** toolbar becomes highlighted. The sketching environment is invoked and the plane is oriented normal to the view i.e., parallel to screen. You will notice that a red color origin is displayed in the center of the screen, indicating the sketching environment. The default screen appearance of the sketching environment of SolidWorks is shown in Figure 4.

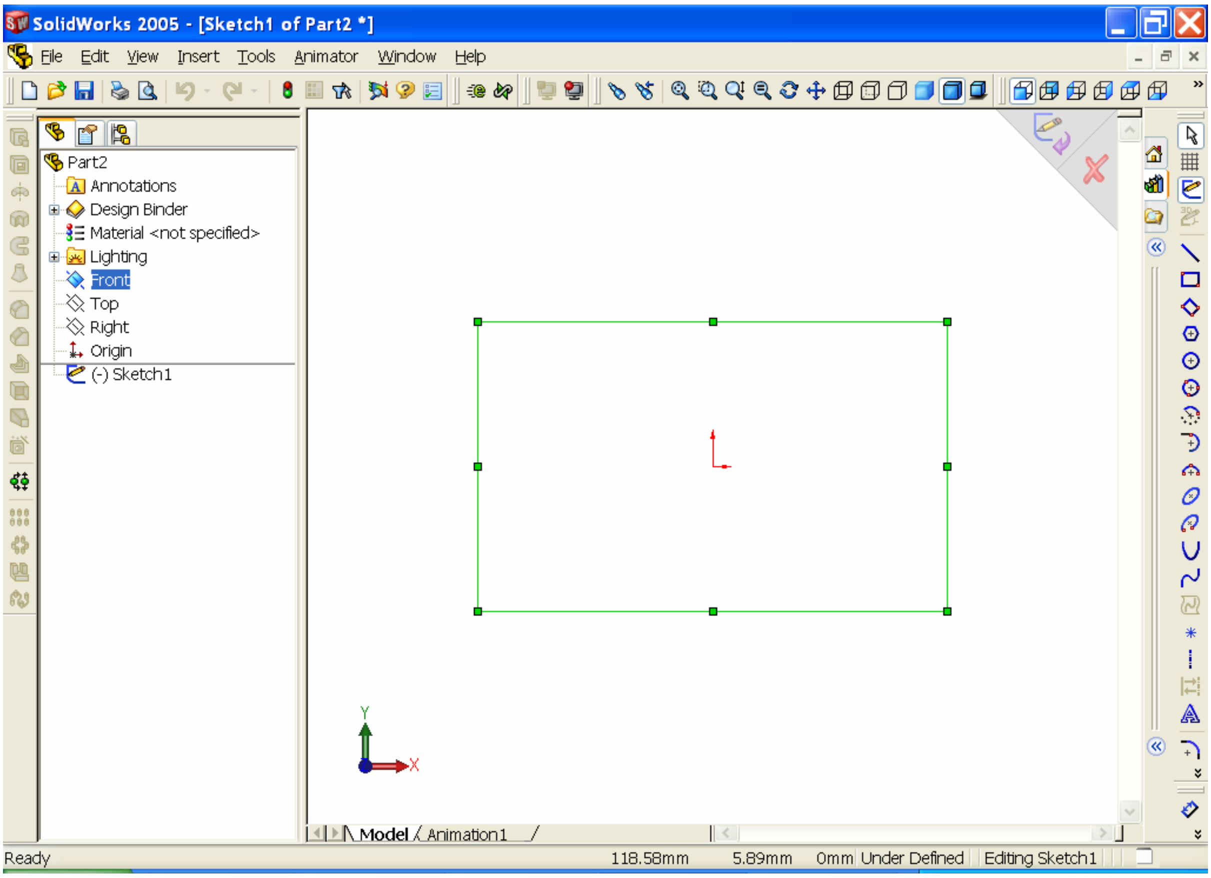


Figure 4 Starting a new sketch.

Setting the Units and Grid

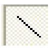
1. Choose **Tools> Options** from the menu bar to invoke the **System Options -General** dialog box.
2. Choose the **Document Properties** tab. The name of the dialog box is changed to the **Document Properties -Detailing** dialog box.
3. Select the Units option from the area on the left to display the options related to linear and angular units.
4. Select **millimeters** from the drop-down list available in the **Length units** area. Also, select the **Degrees** option from the drop-down list provided in the **Angular units** area.
5. Select **Grid/Snap** from the area on the left. Set the value of the **Major grid spacing** spinner to 100 and the value of the **Minor-lines per major** spinner to 10.
6. Select the **Go to System Snaps,** check **Grid** box in the **Sketch Snap** area, if it is cleared. Choose OK to exit the dialog box.

Drawing the Outer Loop of the Sketch

It is a good practice to draw the sketch on one side of the origin, preferably in the first quadrant. This is because while generating the part program for manufacturing the part, you will have a reference for work origin in advance.

The sketch of the model consists of an outer loop, two circles inside the outer loop, and a cavity. Therefore, it will be drawn using the Line and the Circle tools. You will first draw the outer loop and then the inner entities. Note that in the sketcher environment, the lower right corner of the SolidWorks window displays three areas. The first area displays the X, Y, and Z coordinates of the current location of the cursor. These coordinates will be modified as you move the cursor around the drawing area. You will use the coordinate display to draw the sketch of the model.

You will start drawing the sketch from the lower left corner of the sketch and the outer; loop will be drawn using the continuous lines.

1. Click **Line**  button on the Sketch toolbar to invoke the **Line** tool. The arrow cursor will be replaced by the line cursor.
2. Move the cursor in the first quadrant close to the origin. The coordinates of the point are displayed close to the lower right corner of the screen.
3. Press the left mouse button at the point whose coordinates are **10 mm 10 mm 0 mm** and then move the cursor horizontally toward the right.

   You will notice that the symbol of **Horizontal** relation is displayed below the line cursor and the length of the line is displayed above the line cursor. As the length of the first horizontal line at the lower left comer is 10 mm, you will move the mouse until the length of the line is shown as 10 above the line cursor.
4. Press the left mouse button when the length of the line that is displayed above the line cursor shows a value of **10**.

   The first horizontal line is drawn. Because you are drawing continuous lines, the endpoint of the last line is automatically selected as the start point of the next line.
5. Move the line cursor vertically upward. The symbol of **Vertical** relation is displayed below the line cursor and the length of the line is displayed above the line cursor.
6. Press the left mouse button when the length of the line displayed above the line cursor shows a value of **10**.

   A vertical line of length **10 mm** will be drawn and will be displayed in green color. Also, as this is the line that is selected now, the previous line will no more be highlighted and therefore will be displayed in blue color.
7. Move the line cursor horizontally toward the right. Press the left mouse button when the length of the line above the line cursor shows a value of **10**. This draws the next horizontal line of **10 mm** length.
8. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor shows a value of **10**.
9. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor shows a value of **30**.
10. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor shows a value of **10**.
11. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor shows a value of **10**.
12. Move the line cursor vertically downward and press the left mouse button when the length of the line on the cursor shows a value of **10**.
13. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor shows a value of **10**.

14. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **40**.

    The next line that you will draw is an inclined line that makes an angle of 135°. To draw this line, you will move the cursor in a direction that makes an angle of 135°.

    The aligned length of the line is not given for this line. Instead, the delta X and delta Y values are given. Therefore, you will draw a line at this point such that the delta X and delta Y value of this line is **10** mm each. These values can be viewed in the **Line Property Manager**. The last spinners in the **Line PropertyManager** are for the delta values.

15. Move the line cursor such that the line is drawn at an angle of 135-degree. The current angle will be displayed in the spinner above the delta spinners in the **Line PropertyManager**.

16. Press the left mouse button when the delta X value in the **Delta X** spinner shows a value of **10** and the value of delta Y in the **Delta Y** spinner displays a value of **10** in the **Line Property Manager**. At this stage, the length of the line will be displayed as 14.14 above the line cursor and also in the **Length** spinner in the **Line PropertyManager**.

17. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **50**.

    You will notice that some blue and brown inferencing lines are displayed when you move the cursor.

18. Move the line cursor in the direction of 225-degree angle.

    You will notice that two blue inferencing lines are displayed. The first one originates from the start point of the first inclined line and the other one from the start point of the first line of the sketch.

19. Press the left mouse button at the point where the two inferencing lines intersect.

    You will notice that at this point, the length of the line on the line cursor shows a value of 14.14. Also, the delta X and delta Y values in the **Line Property Manager** are displayed as **10** each and the angle is shown as 225-degree.

20. Move the cursor vertically downward to the start point of the first line.

    You will notice that when you move the cursor close to the start point of the first line, a red circle is displayed. Also, the line cursor turns yellow in color and an orange-colored box is displayed below the line cursor. The length of the line shows a value of **40**.

21. Press the left mouse button when the red circle is displayed. Right-click to display the shortcut menu and choose the **Select** option to exit the **Line** tool.

    This completes the sketch of the outer loop. Note that the display of the sketch is small. Therefore, you need to modify the drawing display area such that the sketch fits the screen. This is done using the **Zoom to Fit** tool.

22. Choose the **Zoom to Fit**  button from the **View** toolbar to fit the current sketch on the screen.

The outer loop of the sketch is completed and is shown in Figure 5.

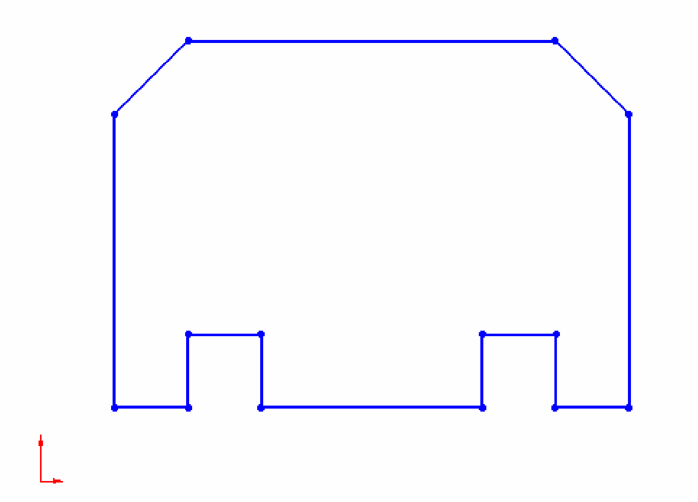


Figure 5 Outer loop of the sketch.

Drawing Circles

The circles will be drawn using the **Circle** tool. You will use the inferencing line originating from the start points and endpoints of the inclined lines to specify the center point of the circles. As mentioned earlier, the cursor in the sketching environment jumps through a distance of 10 mm by default. Therefore, when you move the cursor to define the radius *of* the circle, the minimum value that can be set is 10 mm. But the radius of the circle is 5 mm and to specify this value, the cursor needs to jump only through a distance of 5 mm. This is the reason you need to modify the document settings so that the cursor jumps through a distance of 5 mm.

1. Choose **Tools> Options** from the menu bar to invoke the **System Options -General** dialog box. Choose the **Document Properties** tab.

2. Select the **Grid/Snap** option from the area on the left to display the options related to linear and angular units. Set the value of the **Major grid spacing** spinner to **50** and choose **OK**.

   Setting this value to 50 will force the cursor to jump through a distance of 5 mm instead of 10 mm. Therefore, when you move the cursor now, the values shown on the cursor and the coordinates of the points shown close to the lower right corner of the SolidWorks window will be in the increment of 5 mm.

3. Choose the **Circle**  button from the **sketch toolbar** to invoke the Circle tool.

   As the **Select**  tool was active earlier, the cursor earlier was the arrow cursor. But when you invoke the **Circle**  tool, the arrow cursor will be replaced by the circle cursor.

4. Move the circle cursor close to the lower endpoint of the right inclined line and then move it toward the left. Remember that you will not press the left mouse button at this moment.

   An inferencing line is displayed generating from the lower endpoint of the right inclined line. When you move the cursor toward the left, you will notice that at the point where the cursor is vertically in line with the upper endpoint of the right inclined line, another inferencing line is generated from the upper endpoint of the right inclined line. This inferencing line will intersect the inferencing line generated from the lower endpoint of the inclined line.

5. Press the left mouse button at the point where the inferencing lines from both the endpoints of the inclined lines intersect. Now, move the circle cursor toward the left to define a circle.
6. Press the left mouse button when the radius of the circle displayed above the circle cursor shows a value of **5**.
7. Similarly, draw the circle on the left using the inferencing lines generating from the endpoints of the left inclined line. The sketch after drawing the two circles inside the outer loop is shown in Figure 6.
8. Right-click in the drawing area and choose Select to exit the Circle tool.

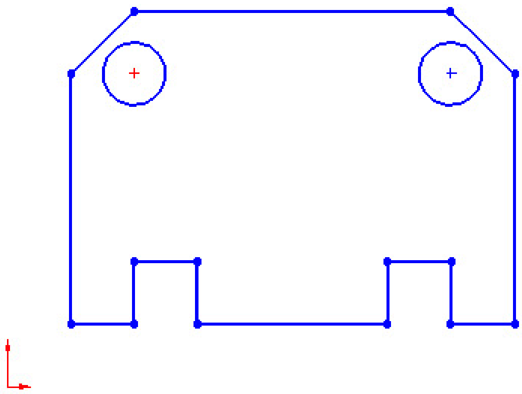


Figure 6 Outer loop of the sketch including circles.

Drawing the Sketch of the Inner Cavity

Next, you will draw the sketch of the inner cavity. To draw the sketch of the inner cavity, you will start drawing with the lower horizontal line.

1. Choose the **Line**  button from the Sketch toolbar. The arrow cursor is replaced by the line cursor.
2. Move the line cursor to a location whose coordinates are **30 mm 25 mm 0 mm**.
3. Press the left mouse button at this point and move the cursor horizontally toward the right. Press the left mouse button when the length of the line above the line cursor shows a value of **30**.
4. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **10**.
5. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **10**.
6. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of **5**.
7. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **10**.
8. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **5**.

9. Move the line horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **10**.
10. Move the line cursor vertically downward to the start point of the first line. Press the left mouse button when the red circle is displayed. The length of the line at this point will show a Value of **10**.
11. Right-click and choose Select from the shortcut menu. This completes the sketch for this Tutorial. The final completed sketch is shown in Figure 7.

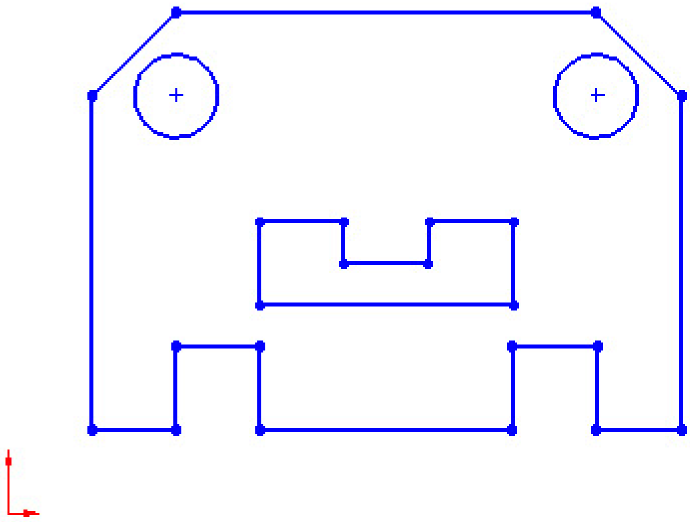


Figure 7 Completed sketch for the model.

Saving the Sketch and model

1. Choose the **Save**  button from the Standard toolbar to invoke the **Save As** dialog box.
2. Enter the name of the document as *tutorial1.sldprt* in the File name edit box and choose the Save button. The document will be saved in the *specified* folder. It is recommended that you create a separate folder for SolidWorks files.

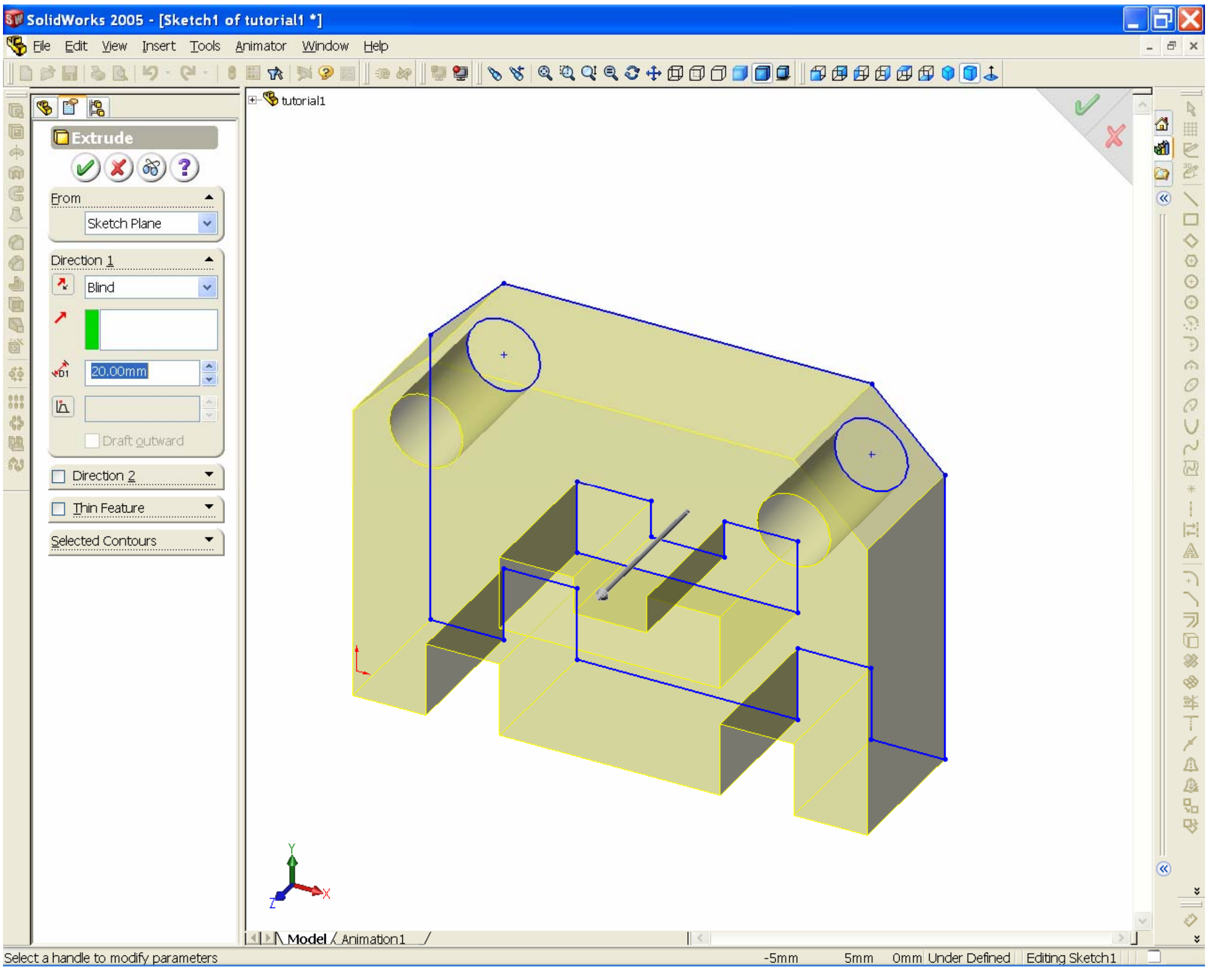


Figure 8

3. Select **Extrude Boss/Base**  on the Features toolbar. The profile changes its orientation to pictorial. The Extrude Property Manager appears as shown in Figure 8.

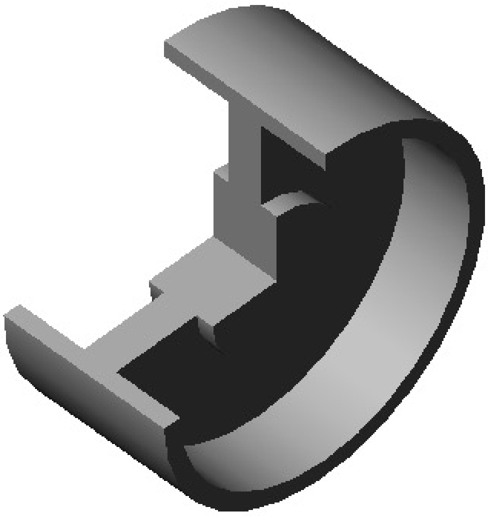
4. In the **Depth**  text box, type **20 mm** as shown in Figure 8. Click **OK**  to finish the extruded base feature.

5. Choose the **Save**  button from the Standard toolbar to save the model.

6. Close the document by choosing **File> Close** from the menu bar.

Module 2b: Basic Sketching II

In this tutorial, you will draw the basic sketch for the revolved solid model shown in Figure 1. The sketch for the revolved solid model is shown in Figure 2.



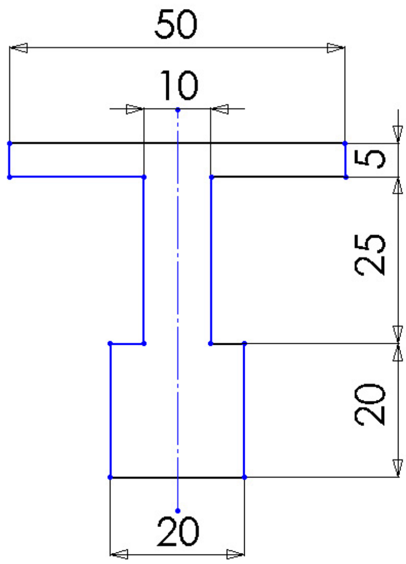


Figure 1 Revolved model for Tutorial 2. **Figure 2 Sketch for the revolved model.**

The steps that will be followed to complete this tutorial are listed below.

Start a new part document.

Maximize the part document and then switch to the sketching environment.

Modify the settings of the snap and grid so that the cursor jumps through a distance of 5 mm instead of 10 mm.

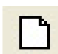
Draw the sketch of the model using the **Line** tool, refer to Figure 3.

Save the sketch and then close the document.

Create a 3-D Model using Revolved Boss/Base Feature.

Save the model and close the document.

Starting a New Document

1. Choose the **New**  button from the Standard toolbar to invoke the New SolidWorks Document dialog box.

2. The **Part** 

button is chosen by default in the New SolidWorks Document dialog box. Choose OK.

   A new SolidWorks part document is opened. But the part document window is not maximized in the SolidWorks window.

3. Choose the **Maximize**  button available on the upper right corner of the part document window to maximize the docun1ent window.

As mentioned earlier, when you open a new part document, the part modeling environment is active by default. But because you first need to draw the sketch of the revolved model, you need to invoke the sketching environment.

4. Select the Front plane. Choose the **Sketch** button from the Standard tool bar.

   A red color origin is displayed and the **Sketch Toolbar** is displayed. Also, the confirmation corner is displayed with the **Exit Sketch** and the **Delete Sketch** options on the upper right corner of the drawing area. This suggests that the sketching environment is activated.

Modifying the Snap and Grid Settings and the Dimensioning Units

Before you proceed with drawing the sketch, you need to modify the grid and snap settings so that you can make the cursor jump through a distance of 5 mm instead of 10 mm, which is the default value.

1. Choose **Tools> Options** from the menu bar to invoke the **System Options-General** dialog box. Choose the **Document Properties tab**.
2. Select the **Grid/Snap** option from the area on the left to display the options related to linear and angular units. Set the value of the **Major grid spacing** spinner to **50**. Make sure the value of the **Minor-lines per major** spinner is **10**. Choose **OK** to close the dialog box.

   The coordinates close to the lower left comer of the SolidWorks window will show an increment of 5 mm instead of the default increment of 10 mm when you exit the dialog box.
3. Make sure the **Snap to points** check box in the **Snap** area is selected.

   If you selected a unit other than millimeter to measure the length while installing SolidWorks, you need to change the unit for the current drawing.
4. Select the **Units** option from the area on the left of the **Document Properties - Grid/Snap** dialog box.
5. Select millimeters from the drop-down list available in the **Length units** area and **Degrees** from the drop-down list available in the Angular units area. Choose the **OK** button after making the necessary settings.

Drawing the Sketch

As evident from Figure 2, the sketch will be drawn using the **Line** tool. You will start drawing the sketch from the lower left corner of the sketch.

1. Choose the **Line** button from the Sketch toolbar. The arrow cursor will be replaced by the line cursor.
2. Move the line cursor to a location whose coordinates are **40 mm 0 mm 0 mm**. An inferencing line originating from the origin is displayed.
3. Press the left mouse button down at this point and move the cursor horizontally toward the right. Press the left mouse button again when the length of the line above the line cursor shows a value of **20**.
4. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **20**.

5. Move the cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **5**.
6. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **25**.
7. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor displays a value of **20**.
8. Move the line cursor vertically upward and press the left mouse button when the length of the line on the line cursor displays a value of **5**.
9. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **50**.
10. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of **5**.
11. Move the line cursor horizontally toward the right and press the left mouse button when the length of the line on the line cursor displays a value of **20**.
12. Move the line cursor vertically downward and press the left mouse button when the length of the line on the line cursor displays a value of **25**.
13. Move the line cursor horizontally toward the left and press the left mouse button when the length of the line on the line cursor displays a value of **5**.
14. Move the line cursor vertically downward to the start point of the first line. Press the left mouse button when the red circle is displayed.

    The length of the line at this point will be **20 mm**.
15. Right-click and choose **Select** from the shortcut menu.

    The sketch is completed but does not fit the screen. Therefore, you need to modify the display area such that the sketch fits the screen.
16. Choose the **Zoom to Fit**  button from the View toolbar to fit the sketch on the screen. The completed sketch for this Tutorial is shown in Figure 3.

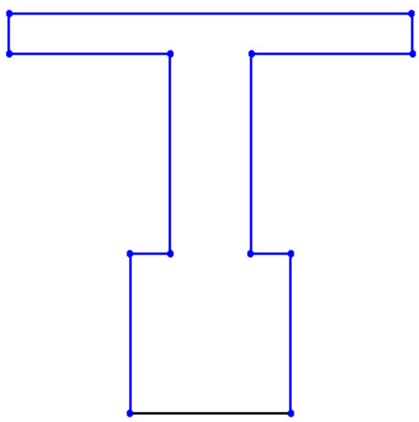
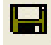


Figure 3 Base sketch for the revolved model

Saving the Sketch

1. Choose the **Save**  button from the Standard toolbar to invoke the Save As dialog box.
2. Enter the name of the document as *tutorial2.sldprt* in the File name edit box and choose the Save button.

The document will be saved in the *selected* folder.

Creating Revolved model

To create a 3-D Model from the Base sketch shown in Figure 3, we will first draw an axis of revolution and then revolve the sketch around the axis.

1. Choose the **Centerline** button from the Sketch toolbar.
2. Move the line cursor to a location whose coordinates are **35 mm 0 mm 0 mm**. An inferencing line originating from the origin is displayed.
3. Press the left mouse button down at this point and move the cursor horizontally toward the right. Press the left mouse button again when the length of the line above the line cursor shows a value of **30**.
4. Right-click and choose **Select** from the shortcut menu.
5. Select **Revolved Boss/Base**  on the Feature toolbar. The Base-Revolved PropertyManager appears as shown in Figure 4. The **One-Direction** is the default type of the revolved feature. The **Angle**  box is filled with **360 deg** automatically.

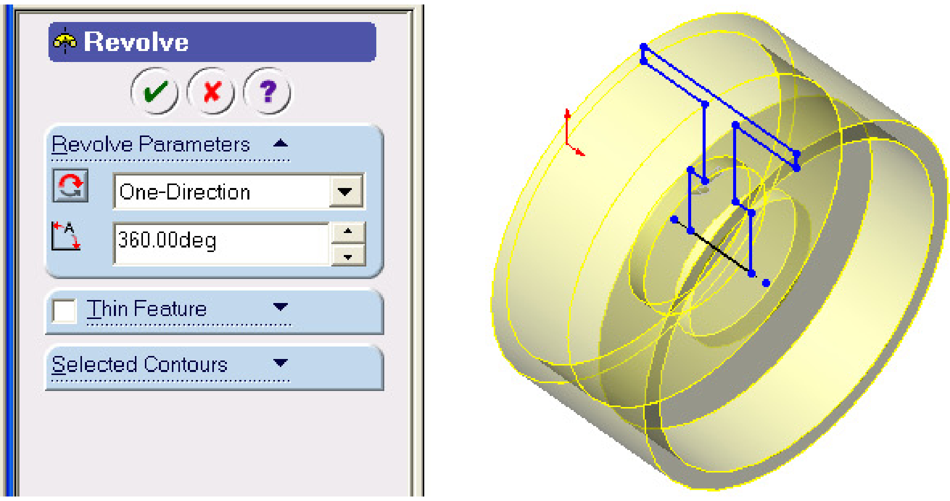
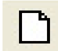


Figure 4 Base- Revolved PropertyManager.

6. Clock **OK**  to accept default setting to create the wheel model shown in Figure 1.
7. **Save** and **close** the file.

An alternate approach to draw the Base sketch shown in Figure 2.

In SolidWorks we may complete the sketch following different approaches. The same sketch shown in Figure 2 will be drawn using another approach.

1. Choose the **New**  button from the Standard toolbar to invoke the New SolidWorks Document dialog box.

2. The **Part** 

button is chosen by default in the New SolidWorks Document dialog box. Choose OK.

3. Select the Front plane. Choose the **Sketch**  button from the Standard tool bar.

   A red color origin is displayed and the **Sketch Toolbar** is displayed. Also, the confirmation corner is displayed with the **Exit Sketch** and the **Delete Sketch** options on the upper right corner of the drawing area. This suggests that the sketching environment is activated.

4. Click **Centerline** on the Sketch toolbar and draw a vertical line approximately **60 mm** in length starting at the origin as shown in Figure 5.

Figure 5 Centerline for the Base sketch.

5. Click **Line**  and complete one side of the sketch starting at the origin as shown in Figure 6. All line segments should be Horizontal or Vertical.

   Do not worry about the sizes of the lines. We will adjust them later.

6. Select centerline and all other lines by making a window around the sketch as shown in Figure 7.

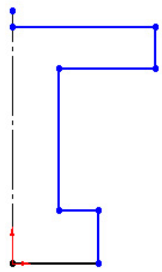


Figure 6 Right half of the sketch.

7. Click Sketch Mirror  on the Sketch toolbar. Sketch profile is complete and is shown in Figure 8.

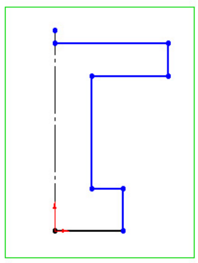


Figure 7 All line segments selected.

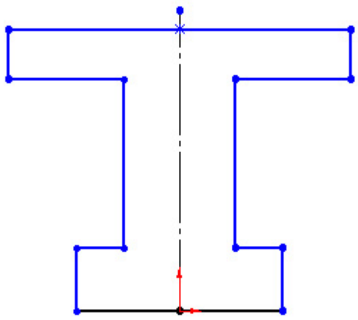


Figure 8 Sketch profile without sizes.

Now we will adjust the sizes of the sketch.

8. Select **Dimension**  on the Sketch Relation toolbar.
9. Click the bottom line. Move the mouse below the bottom line. Click to place the dimension.
10. Change the dimension to **20 mm**.
11. Click the lower right line. Move the mouse to the right. Click to place the dimension.
12. Change the dimension to **20 mm**. (Figure 9).
13. Click the middle vertical lines on the right of the center line then click the similar vertical line on the left of the centerline. Move the mouse upward. Click to place the dimension.

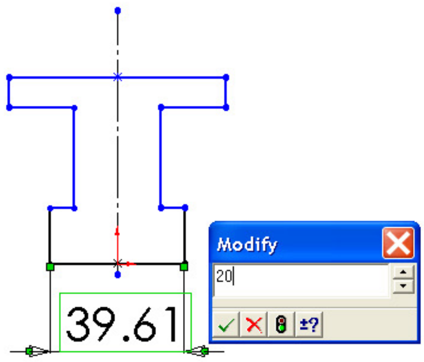


Figure 9 Editing dimension.

14. Change the dimension to **10 mm.**
15. Similarly place the remaining dimensions. The sketch should look like as shown in Figure 10.

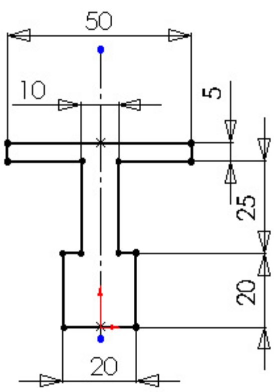


Figure 10 Final sketch with sizes.

16. Choose the **Centerline** button from the Sketch toolbar and draw a horizontal line passing through the origin and extending in both sides of the sketch as shown in Figure 11.

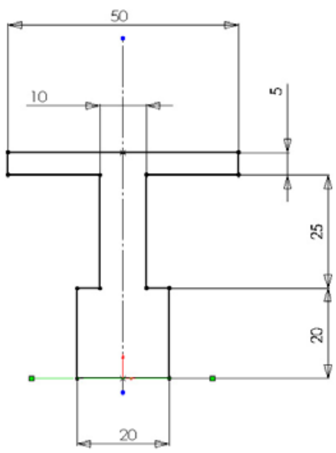


Figure 11 Base sketch with Axis of revolution.

17. Select **Revolved Boss/Base** on the Feature toolbar. The Base-Revolved PropertyManager appears as shown in Figure 12. The **One-Direction** is the default type of the revolved feature. The **Angle** box is filled with **360 deg** automatically.

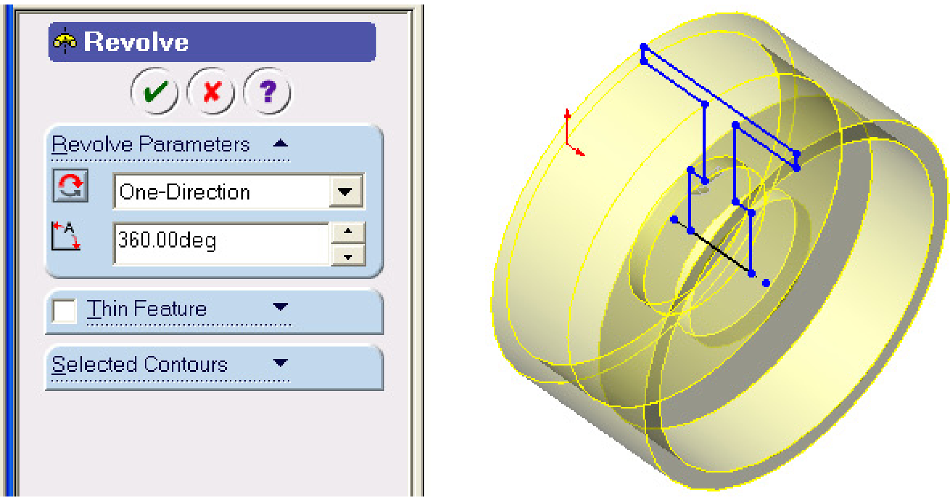


Figure 12 Base- Revolved PropertyManager.

18. Clock **OK** to accept default setting to create the wheel model shown in Figure 1.
19. **Save** and **close** the file.

Module 3: Extruded Feature; V-Block

In order to demonstrate how to create extruded features step by step, a V-Block model is used in this tutorial. Its geometry and dimensions are given in Figure. 1.

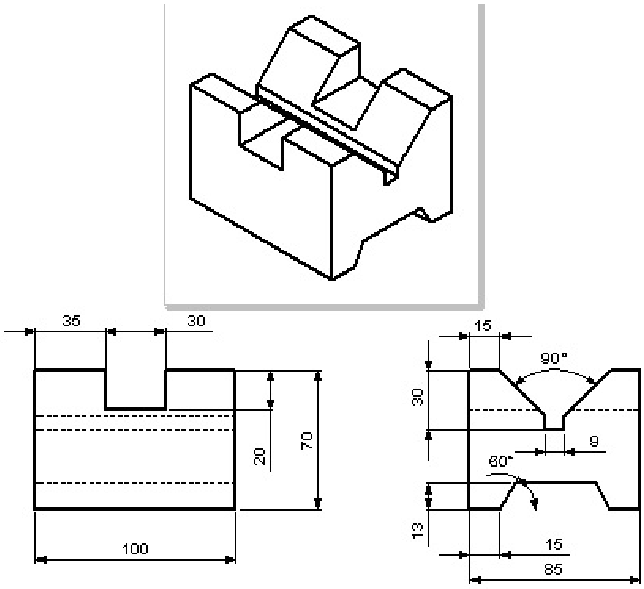


Figure 1. V-Block

Modeling Procedure

Before a model is created, a strategic plan for modeling should be done. The following points should be considered:

- Which view of the part best conveys its shape?
  The ***base feature*** is usually the most prominent feature in that view.
- What is the most important feature of your part?
  Create these features early in the modeling process so you can use them for creating subsequent features.
- Can the origin and coordinate planes be used to modeling advantage?

After the above considerations, the V-Block can be created from the following steps as shown in Figure2.

1) Create a **100mm x 85mm x 70mm** block using the **Extrude Boss/Base**.
2) Cut a **30mm x 20mm x 85mm** Rectangular Slot using **Extrude Cut**.
3) Cut a **V-slot** using **Extrude Cut**.
4) Cut a **Trapezoid slot** using **Extrude Cut**.

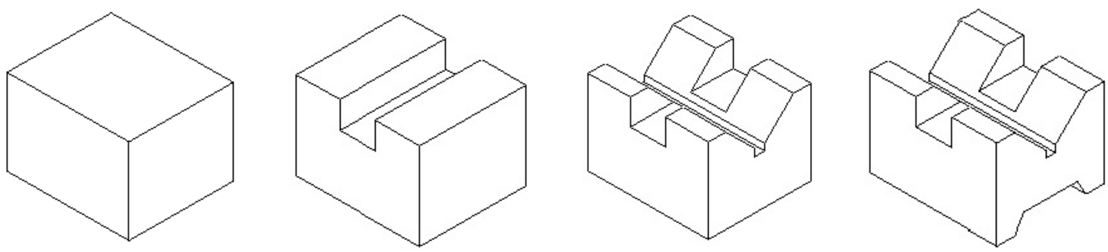
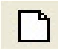


Figure 2 V-Block Modeling steps.

Start Solid Works and Set up Properties

1) Start Solid Works.

2) Click **New**  on the Standard toolbar, or select **File**, **New** on the menu bar.

3) Select **Part** 

from the Tutorial tab in the dialog box and then click **OK**.

4) Click **File**, **Properties** on menu bar.

5) Fill in the necessary properties in the dialog box, and click **OK** to accept the properties and close the dialog box.

6) Select **Tools**, **Options** on the menu bar to open the Options dialog box.

7) Select the **Document Properties tab**, and click **Unit** in the properties tree text box.

8) Select **Millimeter** and set the decimal places to **2** as shown in Figure 3. The available units are Millimeters, Centimeters, Meters, Inches, Feet, Feet, Inches, etc.

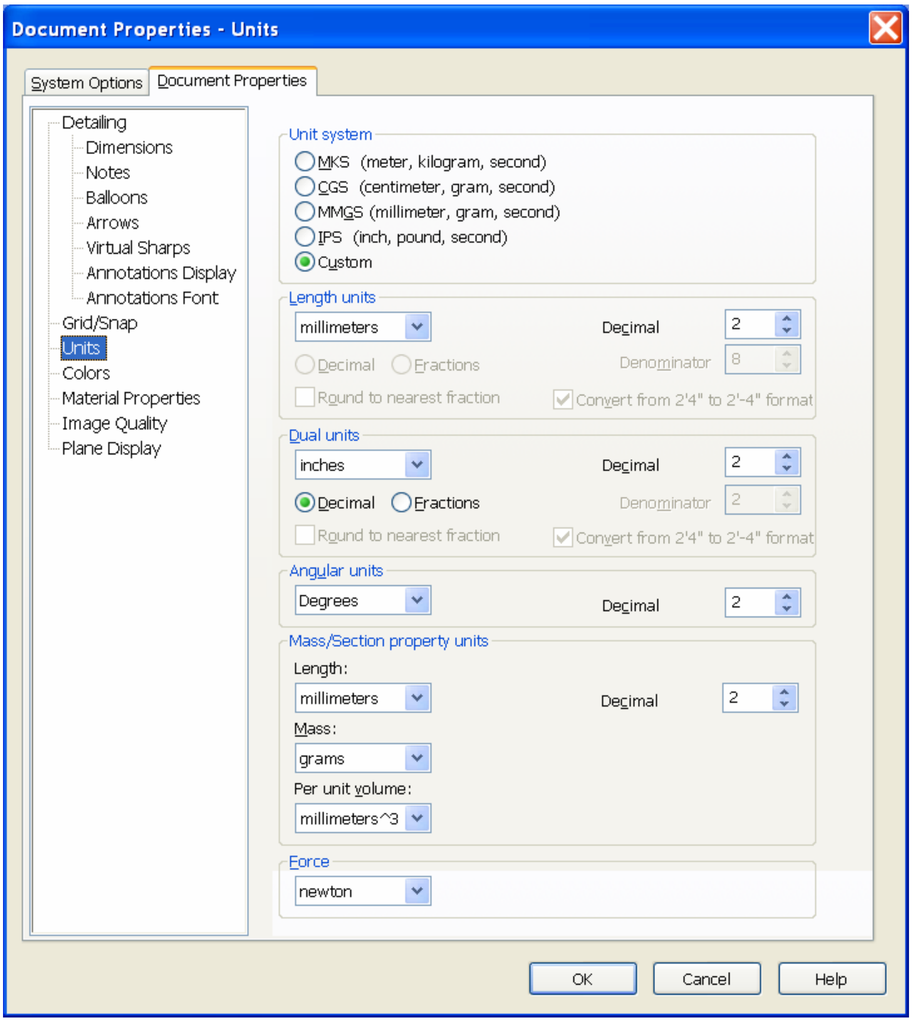


Figure 3 Units selection

9) Click **Material Properties** in the properties tree box. Change the density to **0.0078 g/mm^3** as shown in Figure 4.

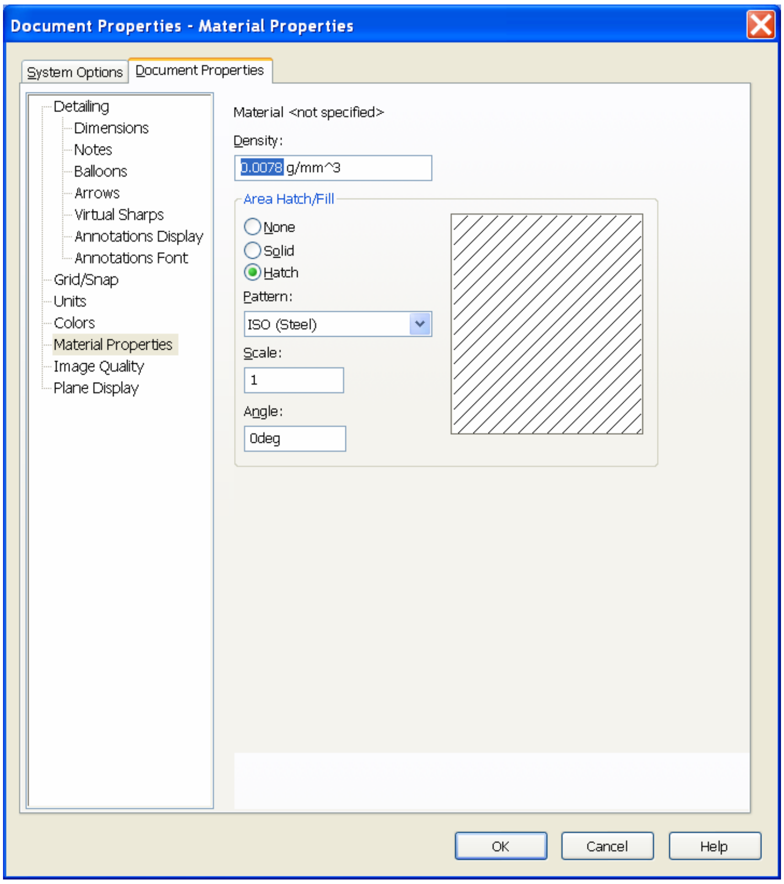


Figure 4 Material properties

10) Click **Grid/Snap** in the properties tree box. Check the **Display grid** check box in the grid frame to display the grid in the sketch environment as shown in Figure 5.

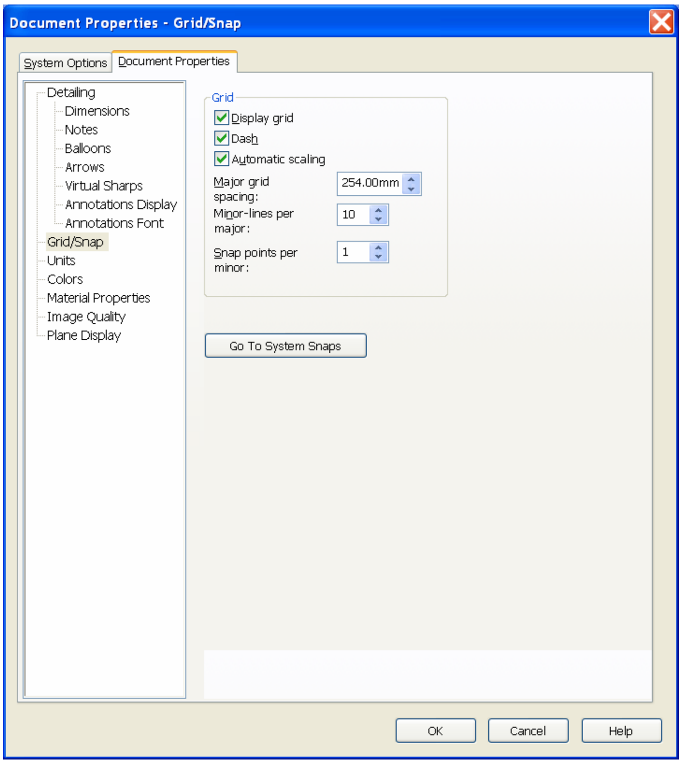


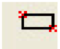
Figure 5 Grid/Snap

11) Click **OK** to accept the changed properties and close the dialog box.

Create Extruded Base Feature

1) Click **Sketch**  on the Sketch toolbar.

2) Bring cursor closer to **Front** plane, when highlighted, Click on it to open a new sketch on **Front** Plane.

3) Select **Rectangle**  on the Sketch Tools toolbar.

4) Click the origin as one corner of the rectangle.

5) Move the mouse up and right, and click to locate the other corner of the rectangle. A rectangle is created. The left vertical line and the bottom horizontal line change the color to black (black is the default color. It could be a different color) because the left lower corner is coincident with the origin, and a coincident relation is applied between the origin and the lower left corner automatically as shown in Figure 6. The top horizontal and right vertical lines are blue (default color) because they need some dimensions to define them.

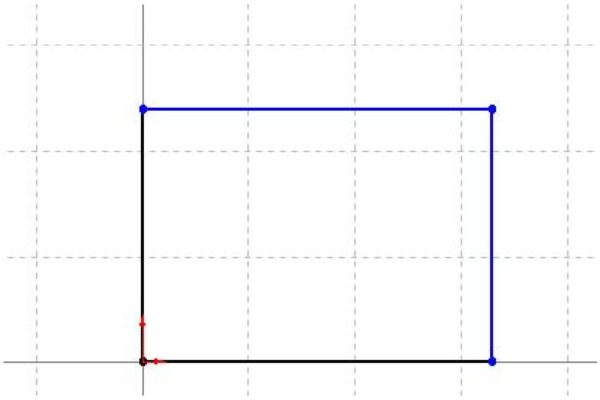


Figure 6 Partially constrained Rectangle

6) Select **Dimension**  on the Sketch Relation toolbar.

7) Click the bottom line. Move the mouse below the bottom line. Click to place the dimension.

8) Change the dimension to **100 mm**.

9) Click the right line. Move the mouse to the right. Click to place the dimension.

10) Change the dimension to **70 mm.** The sketch is fully constrained, and turns to black (default color) as shown in Figure 7.

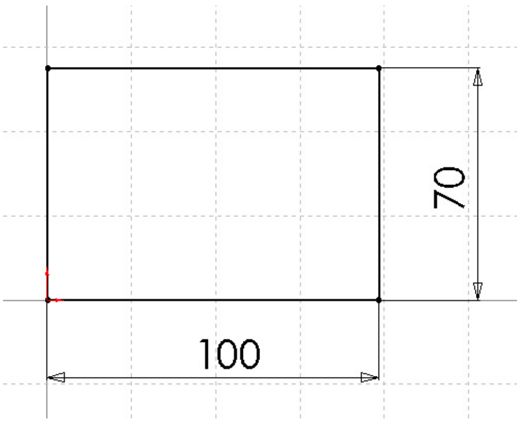


Figure 7 Base sketch of the Block

11) Select **Extruded Boss/Base**  on the Features toolbar. The profile changes its orientation to isometric. The PropertyManager appears.

12) Move the mouse to the left side of the sketch, and click to accept the extrude direction.

13) In the **Depth**  text box, type **85 mm** as shown in Figure 7. Click **OK**  to finish the extruded base feature.

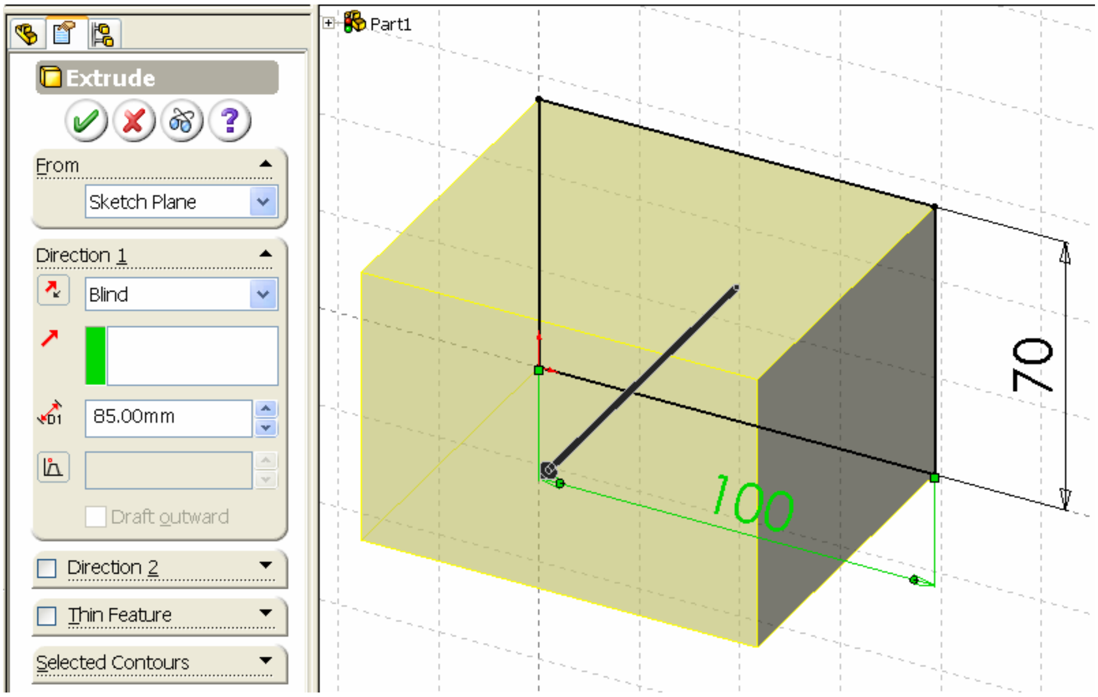


Figure 7 Extruded Base Feature

Cut Rectangular Slot

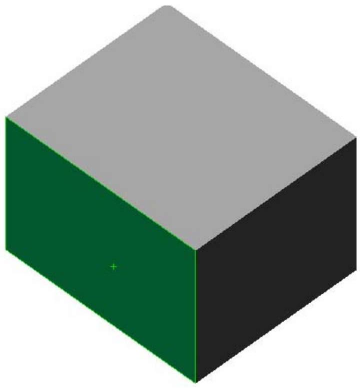


Figure 8 Sketch Plane for Rectangular Slot

1) Select the front surface of the base feature as shown in Figure 8.

2) Select **Sketch**  on the Sketch toolbar to open a sketch on the front surface.

3) Select **Normal To**  on the Standard Views toolbar to make the sketch parallel to the screen. (If the Standard Views toolbar is not displayed, select **View, Toolbar, Standard Views** on the menu bar to open the Standard Views toolbar.)

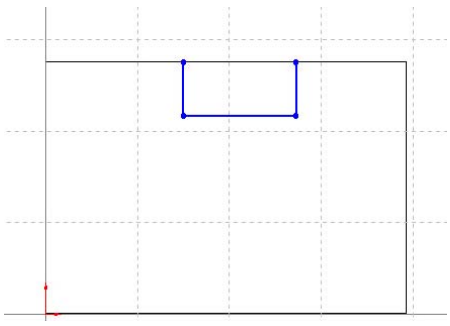
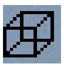


Figure 9 Rectangular Cut Sketch

4) Select **Wireframe**  on the Views toolbar to change the model display to wireframe from the shaded mode.

5) Select **Line**  on the Sketch Tools toolbar to draw 3 lines as shown in Figure 9.

6) Select **Dimension**  on the Sketch Relation toolbar.

7) Dimension the sketch as shown in Figure 10. The sketch is fully constrained and turns to black. if not, apply coincident constraints to the endpoints of the vertical lines with the top edge of the part.

8) Select **Extruded Cut**  on the Feature toolbar.

9) Select **Through All** from the end condition drop down list box in the Cut-Extrude PropertyManager as shown in Figure 11.

10) Click **OK**  to finish the cut.

11) Click **Shaded**  on the View toolbar to change the model to Shaded mode to check the cut.

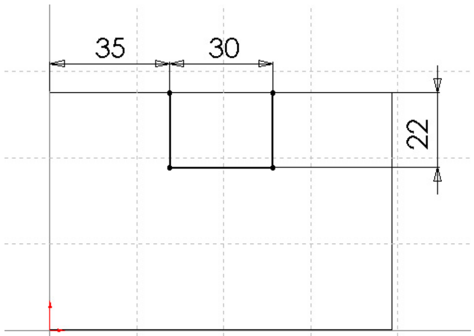


Figure 10 Rectangular Slot Dimensions

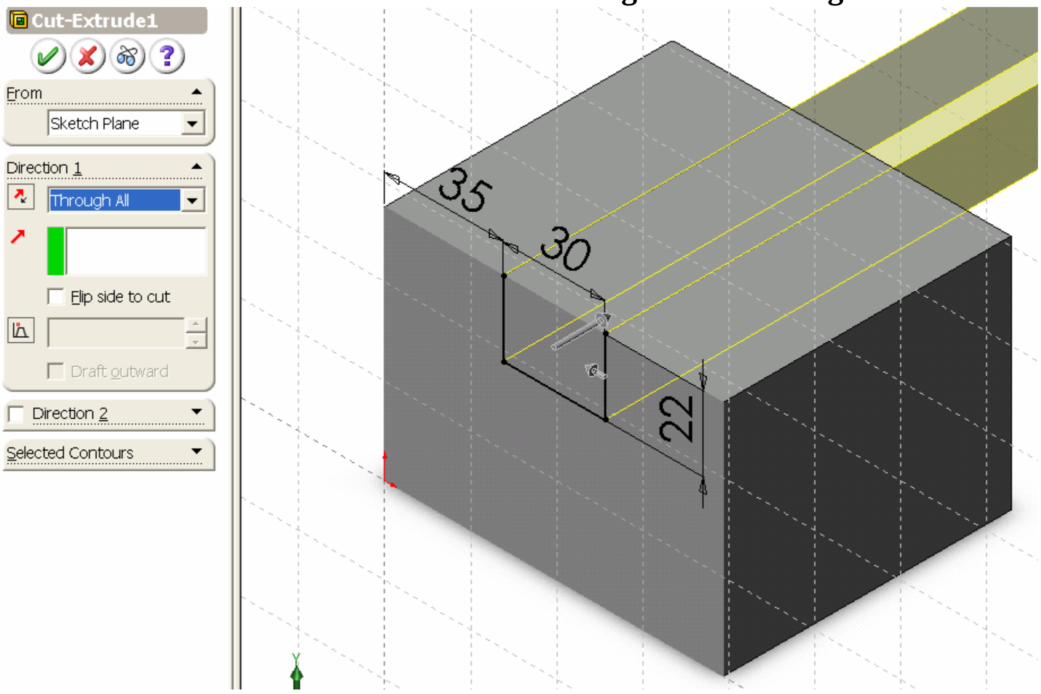


Figure 11 Rectangular Slot Cut

Cut V-Slot

1) Select the right surface of the part as the V-Slot sketch plane as shown in Figure 12.

2) Select **Sketch**  on the Sketch toolbar to open a new sketch.

3) Select **Normal To**  on the Standard Views toolbar to make the sketch parallel to the screen.

4) Select **Line**  on the Sketch Tools toolbar.

5) Draw five lines to form the geometric V-slot shape as shown in Figure 13.

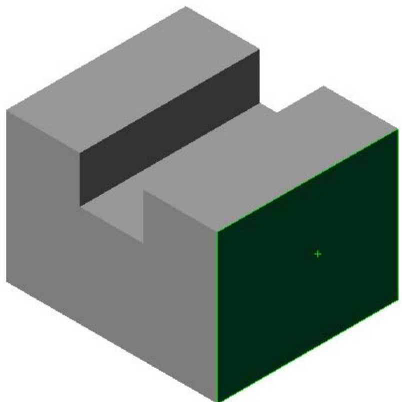


Figure 12 Sketch plane for V-Slot Cut

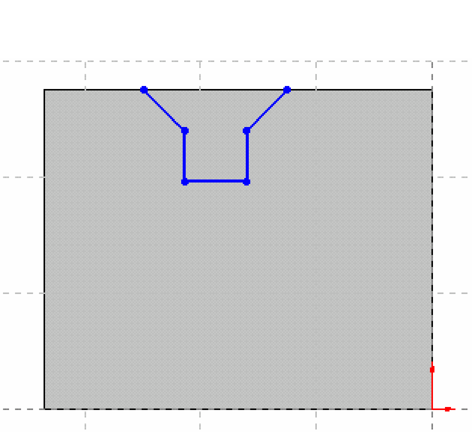


Figure 13 V-Slot Sketch

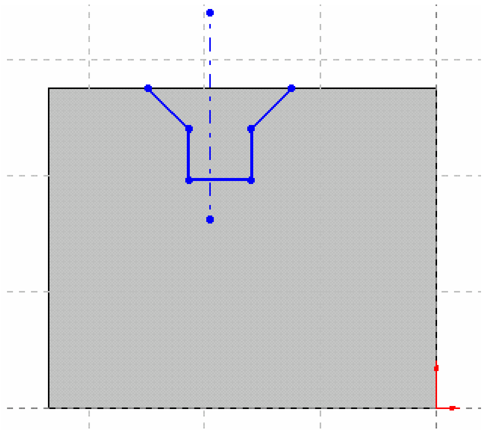


Figure 14 Centerline for V-Slot Sketch

6) Select **Centerline** on the Sketch Tools toolbar.

7) Draw a vertical centerline within the V-slot sketch as shown in Figure 14.

8) Select **Add Relation** on the Sketch Relation toolbar.

9) Select two angled lines and the centerline.

10) Select **Symmetric** in the *Add* Relation PropertyManager as in Figure 15 to make two angle lines symmetric about the centerline.

11) Select two vertical lines and the centerline.

12) Select **Symmetric** in the Add Relation dialog box as shown in Figure 16 to make two vertical lines symmetric about the centerline.

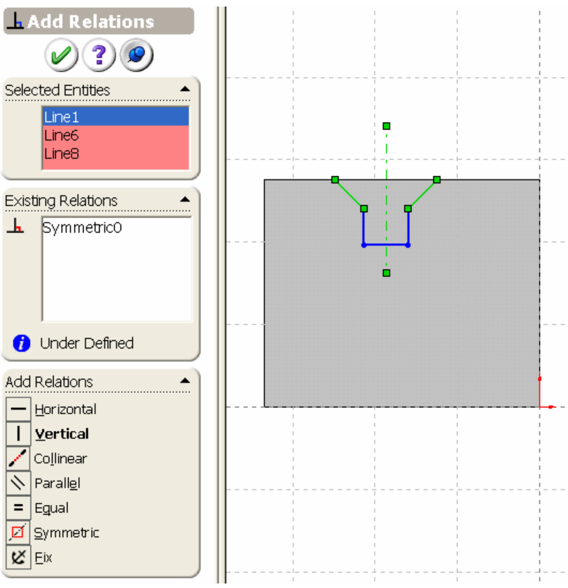


Figure 15 Symmetric Constraints for Two Angled Lines

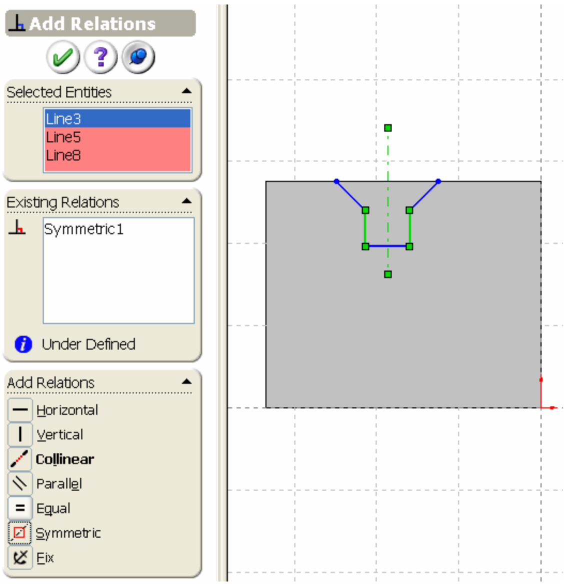
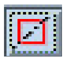


Figure 16 Symmetric Constraints for Two Vertical Lines

13) Select two vertical edge lines on the face and the centerline.

14) Select **Symmetric**  in the Add Relation dialog box as shown in Figure 17 to make centerline at the center of face.

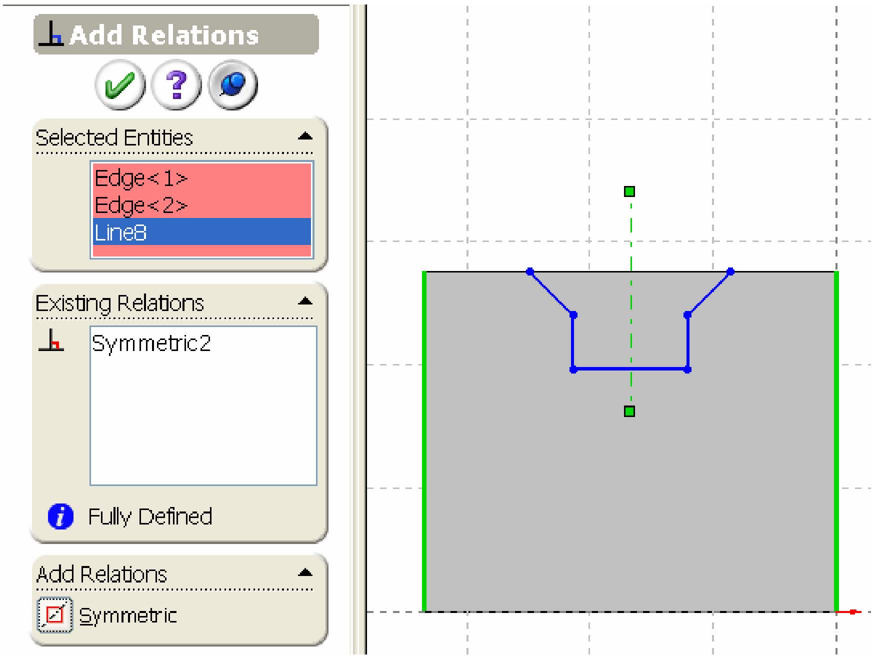


Figure 17 Symmetric Constraints for the centerline at the center of the face

15) Click on the **OK**  button to close the Add Relation PropertyManager.

16) Select **Dimension** on the Sketch Relation toolbar.

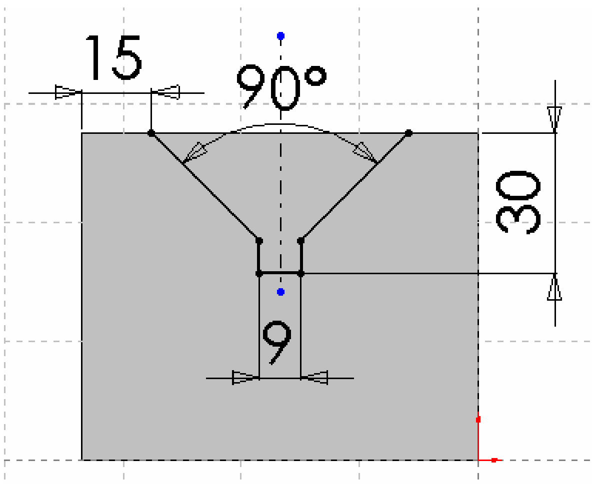


Figure 18 V-Slot dimensions

17) Dimension the sketch as shown in Figure 18. The sketch is fully constrained. If not, apply coincident constraints to the two endpoints of the vertical lines with the top edge of the part. You may also drag the end point of the angled line to make it coincident with the top horizontal edge of the part.

18) Select **Extruded Cut** on the Features toolbar

19) Select **Through All** from the end condition drop down list box in the Cut-Extrude PropertyManger.

20) Uncheck **Direction 2** as shown in Figure 19.

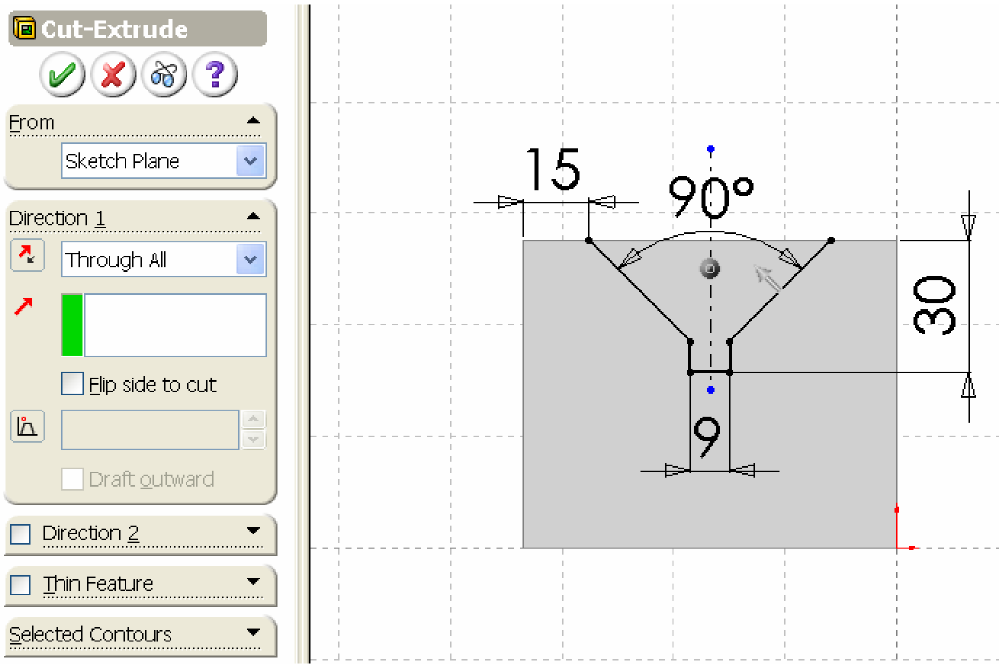


Figure 19 V-Slot Cut.

21) Click **OK** to finish the cut.

21) Click **Rotate View** on the View toolbar, and drag the mouse to rotate the model. Right click and select **Rotate View** from the context menu to end the rotate view command.

22) Select **Isometric** on the Standard Views toolbar to set the part orientation to Isometric.

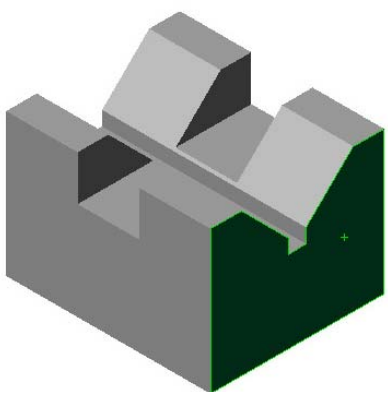


Figure 20 Sketch plane for Trapezoid Slot cut

Cut a Trapezoid Slot

1) Select the same surface as the V-Slot cut as the sketch plane for the trapezoid cut as shown in Figure 20.

2) Select **Sketch** on the Sketch toolbar to open a new sketch.

3) Select **Normal To**  on the Standard Views toolbar to make the sketch parallel to the screen.

4) Select **Line**  from the Sketch Tools toolbar.

5) Draw three lines to form the trapezoid slot sketch as shown in Figure 21.

6) Select **Centerline**  on the Sketch Tools toolbar.

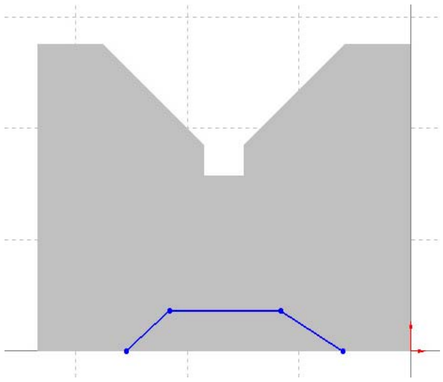
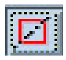


Figure 21 Trapezoid Slot Sketch

7) Draw a vertical centerline starting from the mid-point of the lowest horizontal edge as the symmetric axis for the trapezoid slot.

8) Select **Add Relation**  on the Sketch Relation toolbar.

9) Select the two angled lines and the centerline. Select **Symmetric**  in the PropertyManager as shown in Figure 22.

10) Click **OK**  to close the PropertyManager.

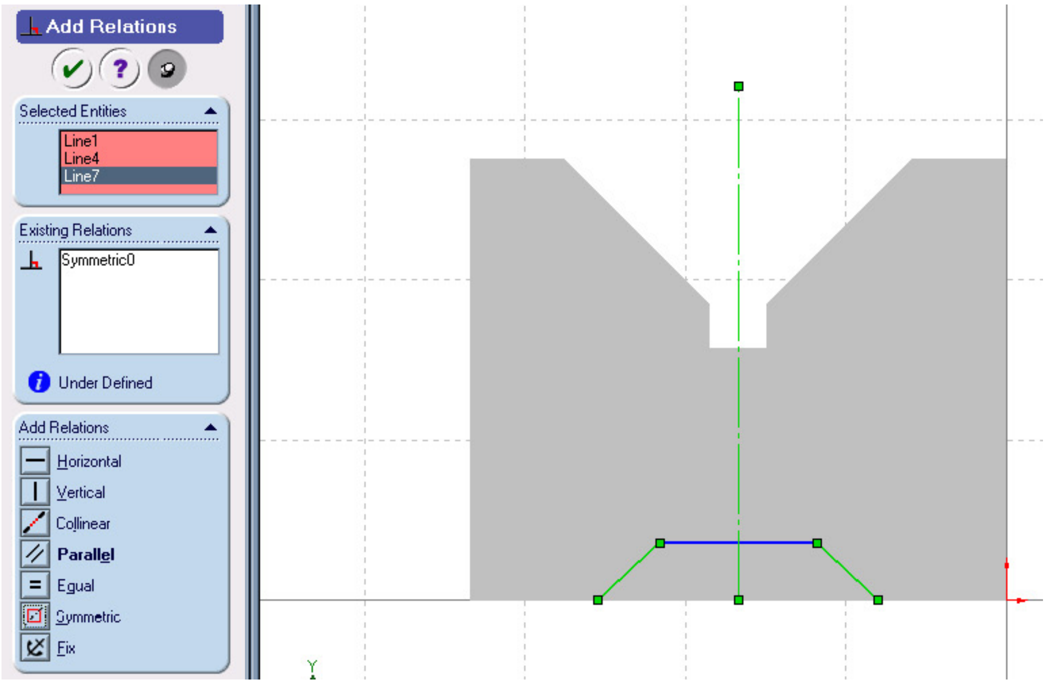


Figure22 Symmetric constraint for Angled lines

11) Select **Dimension**  on the Sketch Relation toolbar.

12) Dimension the sketch as shown in Figure 23.

13) Select **Extruded Cut**  on the Features toolbar.

14) Select **Through All** from the end condition drop down list box in the Cut-Extrude PropertyManger.

15) Check **Flip side to cut** to change the cut direction toward inside.

16) Uncheck **Direction** 2 as shown in Figure 24.

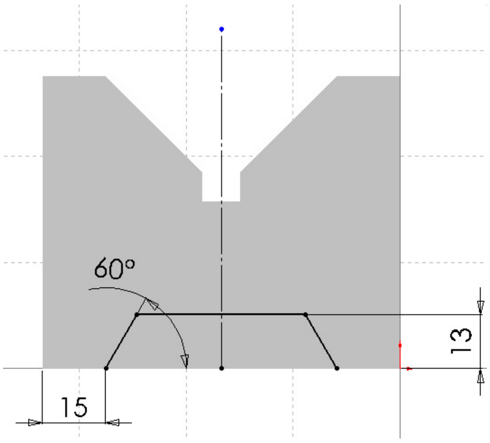


Figure 23 Trapezoid Slot Dimensions

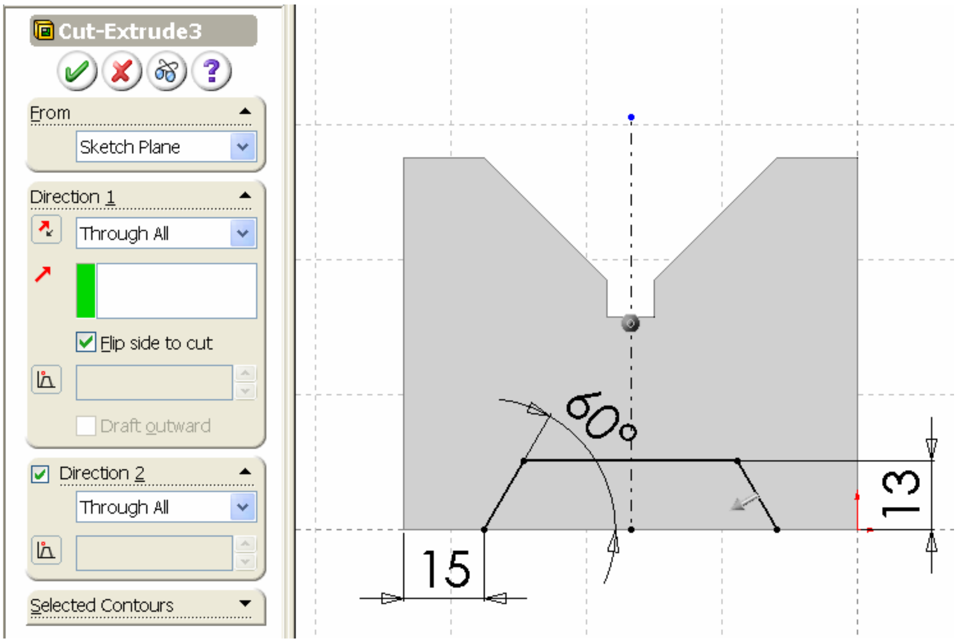


Figure 24 Trapezoid Slot Cut

17) Click **OK**  to finish the cut. The final part is shown as in Figure 25.

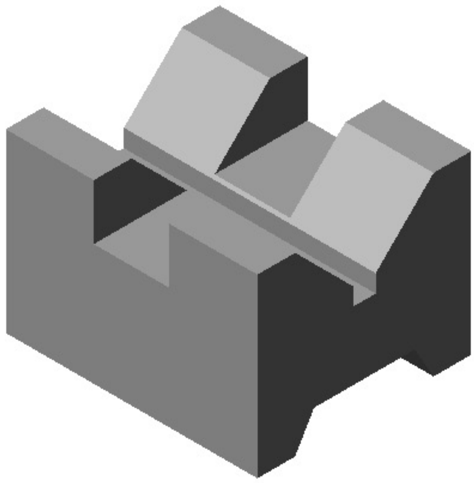


Figure 25 V-Block Model

Change Feature Names and Check Model Mass Properties

Once features are created, they are listed in the ***FeatureManager Design Tree*** as shown in Figure 25 with default names. You may have slightly different default names. Those names could remain as they are, but, it is recommended to change the default names to make the features recognized easily:

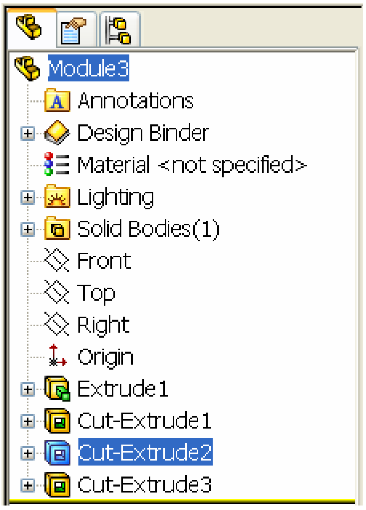


Figure 26 FeatureManager Design Tree

To change the default names:

1) Click **Extrude1** twice (not double click) in the FeatureManager Design Tree.
2) Type ***Block*** as the new name.
3) Click **Cut-Extrude1** twice, and change it to ***Rectangular-Slot.***
4) Click **Cut-Extrude2** twice, and change it to ***V-Slot.***
5) Click **Cut-Extrude3** twice, and change it to ***Trapezoid-Slot.***

The new FeatureManger Design Tree is as shown in Figure 27.

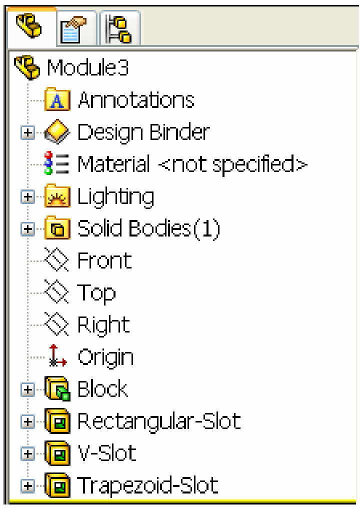


Figure 27 Updated FeatureManager Design Tree

To check the model mass properties:

1) Select **Tools, Mass Properties** on the menu bar.

2) The Mass Properties dialog box appears. The Measurement Options dialog box appears, and the mass properties are calculated automatically as in Figure 28. **Note:** in the dialog box, the mass density is displayed as 0.01 gram per cubic millimeter because the decimal places format is set as 2. It does not influence the mass properties.

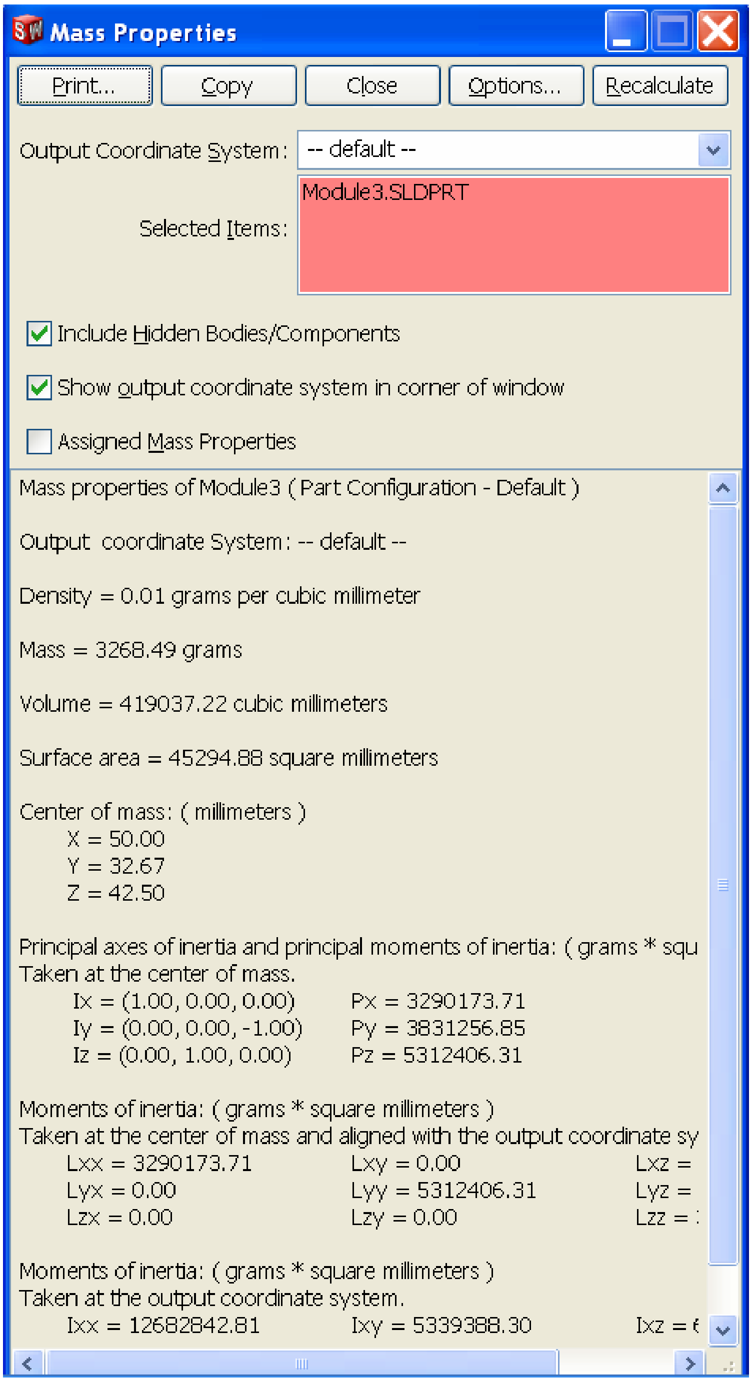
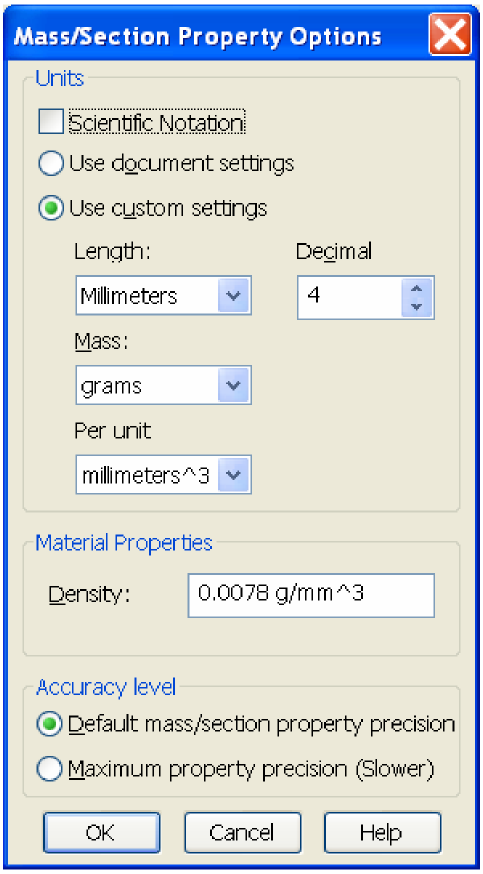


Figure 28 Mass Properties

3) To display the correct value of the mass density, select the **Options** button. The **Mass/Section Property Options** dialog box appears as shown in Figure 29. Change the **Decimal Places** to 4, the mass density is displayed as **0.0078 g/mm^3** in the Density text box as shown in Figure 30.



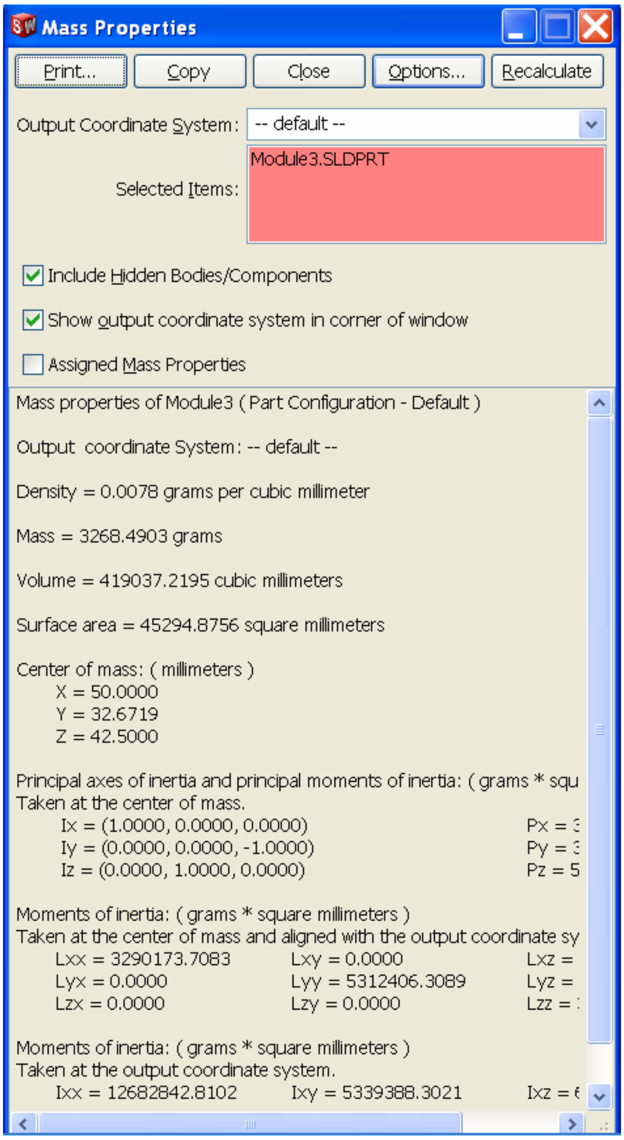


Figure 29 Mass/Section Property Options

Figure 30 Mass Properties

4) Click the **OK** button to close the Options dialog box.

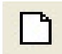
5) Click on the **Print** button to print the mass properties, or click **Copy** to copy them into the clipboard. Then they can be copied to other programs.

6) Click **Close** to close the Mass Properties dialog box.

7) Choose the **Save**  button from the Standard toolbar to save the model.

Module 4: To Construct Housing Body and Add Multiple Features

1. Start the Program  **Desktop Symbol** OR **Start Menu**

2. Select '**New Document**'  from the Welcome window OR Select '**New**' button OR Select File-new OR **Ctrl+N**.

3. Select '**Tutorial Tab**' and then '**Part**' 

4. Select '**Top**' (plane) from the '**Feature Manager Design tree**'.

5. Select '**Normal to**'  from the Standard Views Toolbar.

6. Select '**Sketch**' button  OR Insert – Sketch from the dropdown menu bar at the top of your screen.

7. Select the '**Rectangle**' button . Notice that the cursor changes to a pencil symbol with a rectangle.

8. Place the Rectangle pencil at the lower left of the red origin and click the left button of the mouse <u>**ONCE**</u> **(left click)**.

9. Drag the mouse to the top right of the red origin. (Notice the x-y coordinates changing at the bottom of the screen) Left click once. The size of the rectangle is not important at this stage. The rectangle is now green and the symbol is disconnected from the sketch.

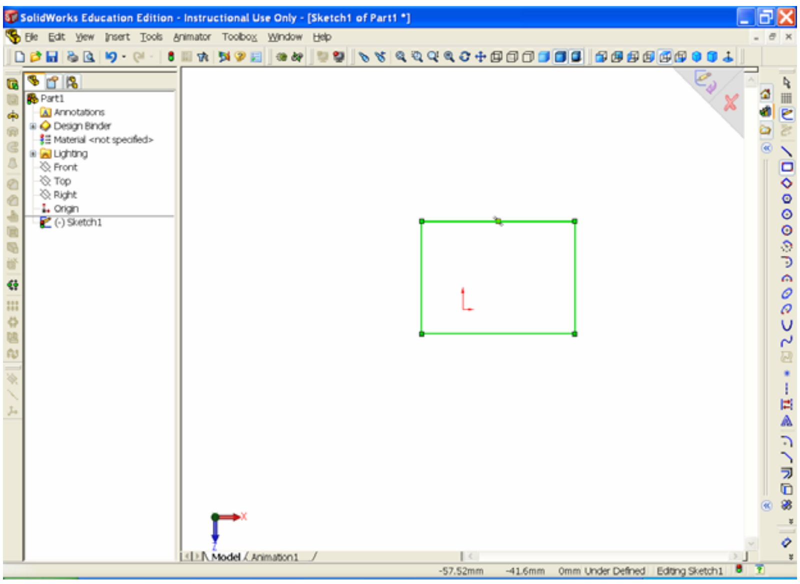


Figure 1

N.B. Look at the bottom right of the screen. As the cursor moves, the x & y coordinates change to indicate the location, 'z' is zero since this is a 2D sketch. The screen also indicates that we are 'Editing Sketch 1' and that the sketch is 'Under Defined'.

10. Choose '**Select**'  from the Sketch Toolbar. The sketched rectangle is now blue and the Rectangle Command is complete.

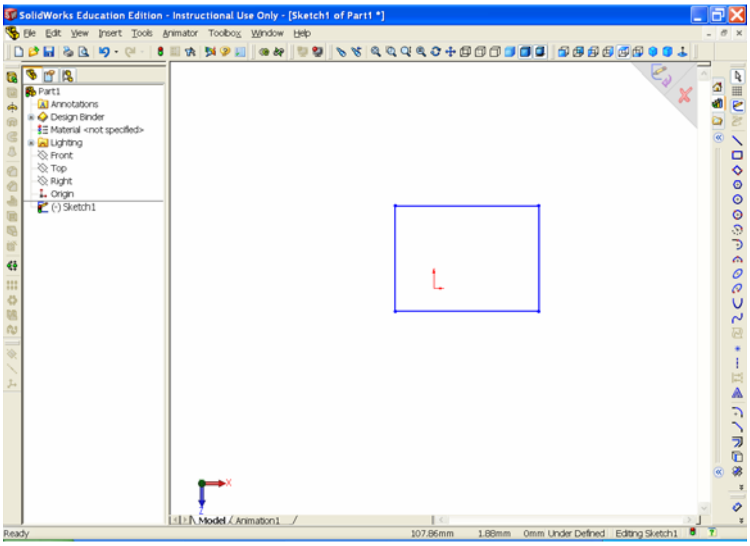


Figure 2

11. Choose '**Smart Dimension**'  from the Sketch Toolbar. The cursor now indicates a dimension symbol.

12. Dimension the sketch as shown in the Figure 11.

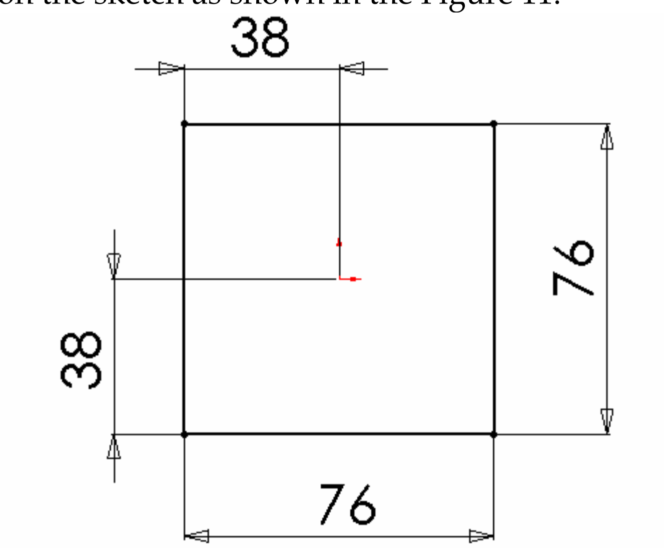


Figure 3

Notice that the sketch is now completely black, that it is 76mm square and that it is centred on the origin. Note also that the box at the bottom right of the screen indicates that the sketch is now fully defined.

13. Choose the select button to end the dimension command.

The sketch can now be transformed into a 3D solid object. Notice the Features Toolbar is now giving access to the Extruded Boss/Base Tool .

14. Choose the '**Extruded Boss/Base**' button .

The Extrude manager opens and the sketch is displayed on screen as an isometric view. A default distance of 10mm is added to the model in the direction of the arrow shown.

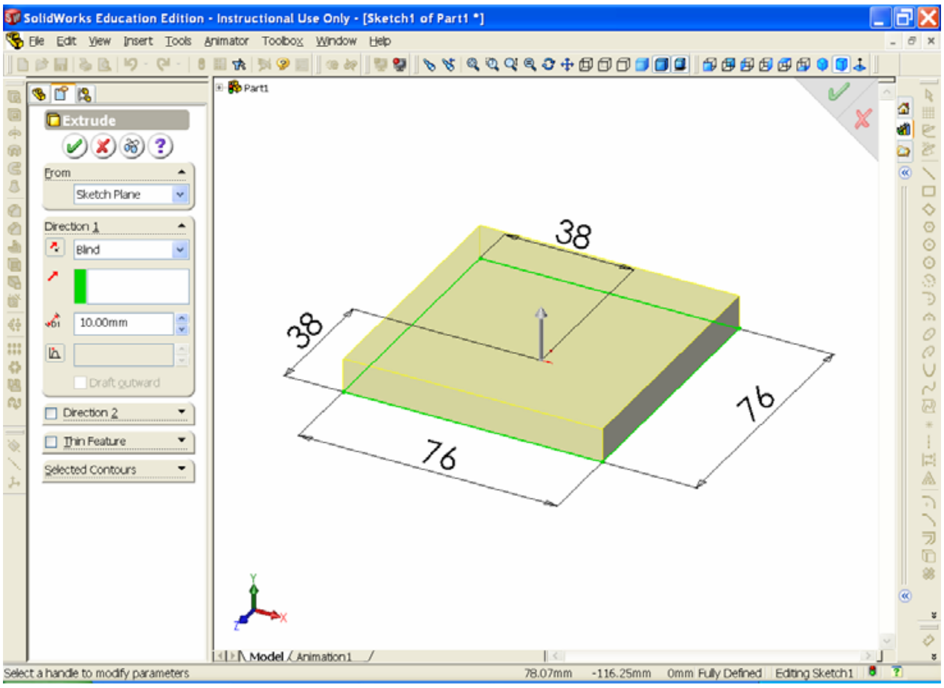


Figure 4

15. Leave the End Condition as '**Blind**'. Use the up arrow at the side of the depth box (d1) to increase the value to 20mm. Notice that the model increases by 10mm in the direction of the arrow on screen. Highlight this 20mm value and use the backspace key on the keyboard to remove.
Replace the 20mm with a new value of 38mm and press the enter key. Once again the model changes to the new value. Select the green tick to accept the model with this distance.

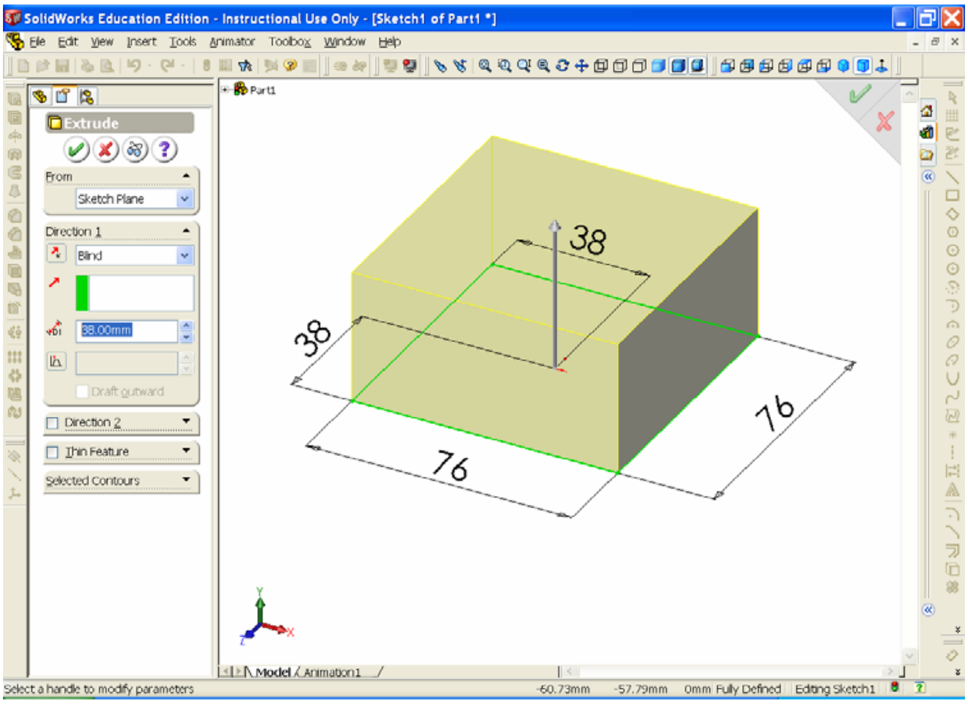


Figure 5

Notice that the Feature Manager Design tree now shows Extrude 1 with a '+' box to its left. Place the cursor on the '+' box and left click to open it. Notice that Sketch 1 is the base of the extruded model.

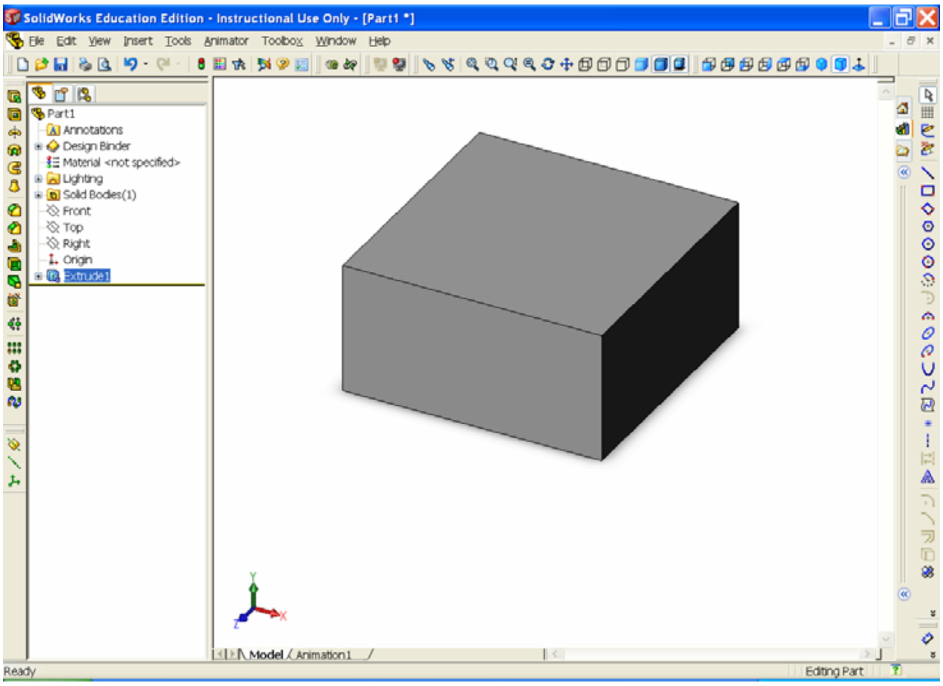


Figure 6

16. Right click on Sketch 1 and choose **Edit sketch**.

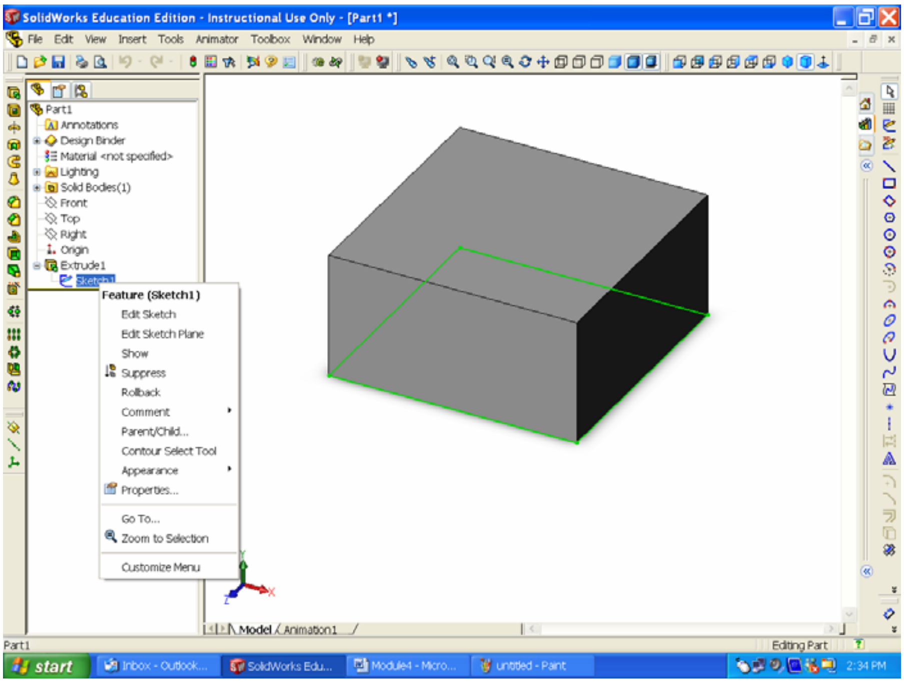


Figure 7

17. Choose ‘**Normal to**’ from the Standard Views Menu.

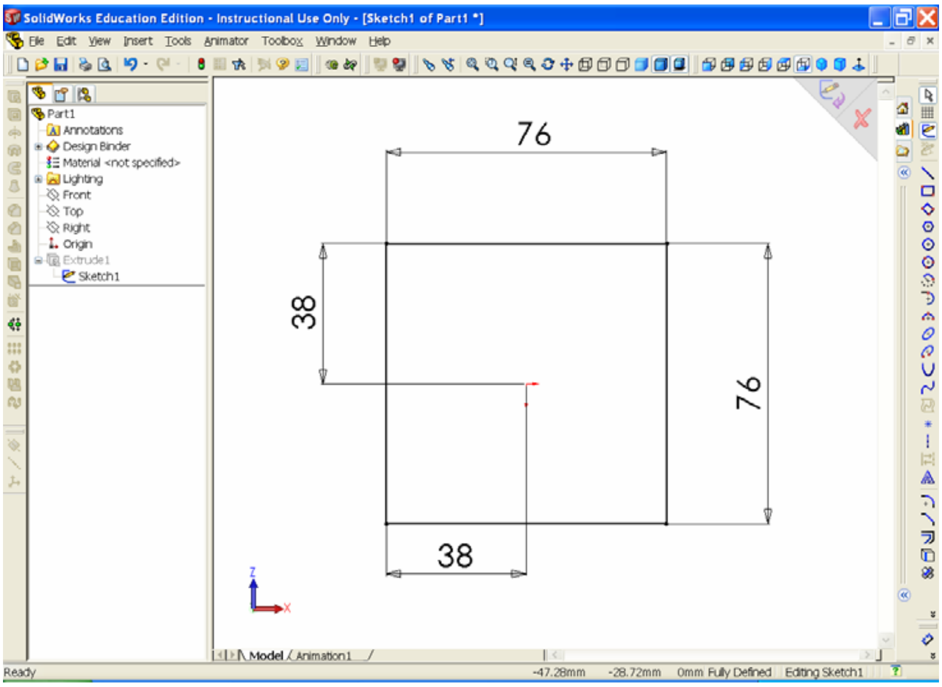


Figure 3

The now familiar returns to the screen and may be edited.

18. Choose ‘**Sketch fillet**’ from the Sketch tools Toolbar.

The sketch fillet property manager opens with a default radius of 10mm.

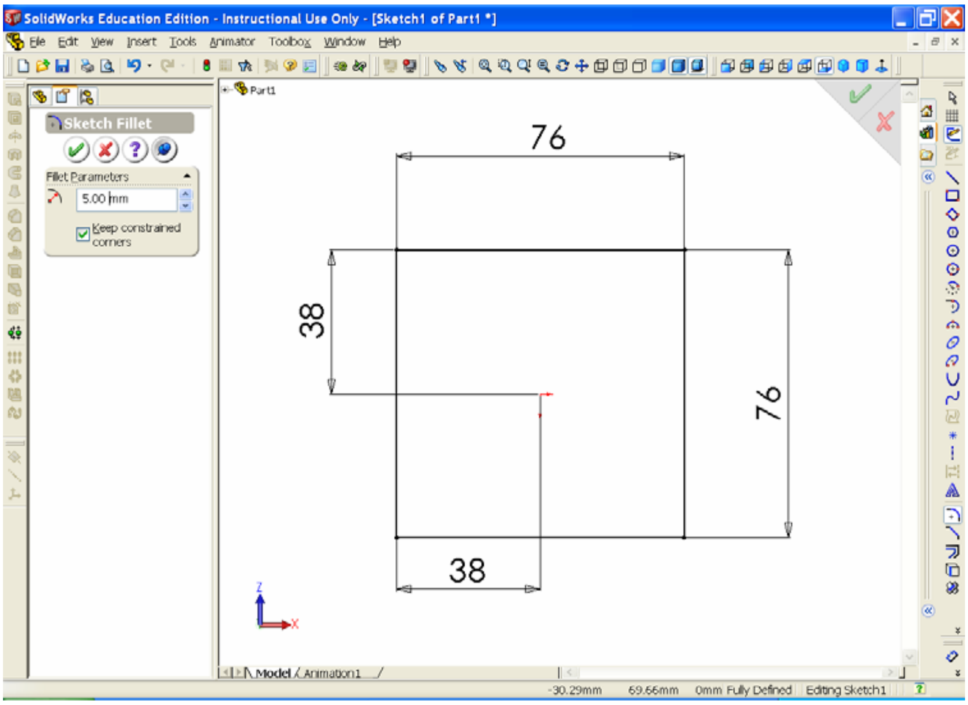


Figure 4

19. Change this radius to 5mm and select each corner of the sketch.

Notice that each corner is highlighted by a red dot as it is identified. Also note that the fillet dimension is added to the sketch on the first fillet only. Ensure that the 'Keep constrained corners' box is checked. Select the green tick ✔ *(OK).*

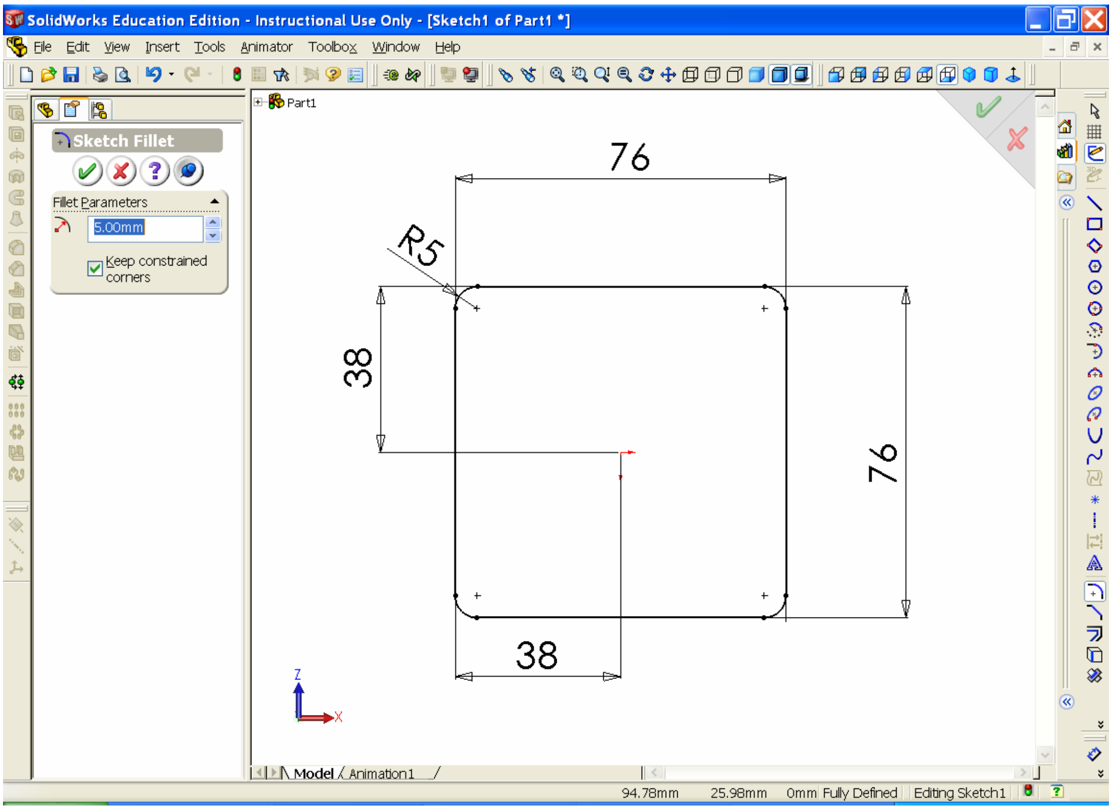


Figure 5

The Feature Manager Design tree now shows a rebuild required symbol to the left of Extrude 1.

20. Left click the '**Rebuild**' button and select '**Isometric'** from the Standard Views menu.

Figure 6

We now need to make the base of the model hollow. To view the base we must rotate the view to see the flat bottom.

21. Choose the '**Rotate view**' button. The cursor now appears like two semi circular arrows following each other. Press the left button of the mouse and hold it down. Still holding the left button down rotate the model so that the bottom surface is visible. Release the mouse button when you are finished rotating the model and press the rotate button once more to end the command.

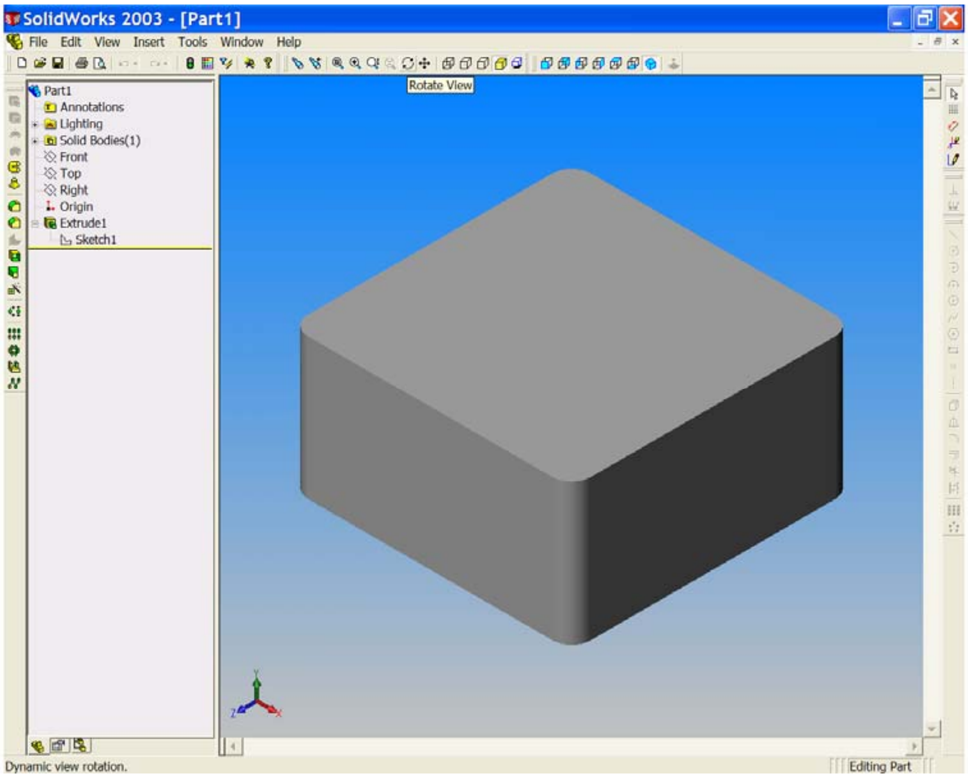


Figure 7

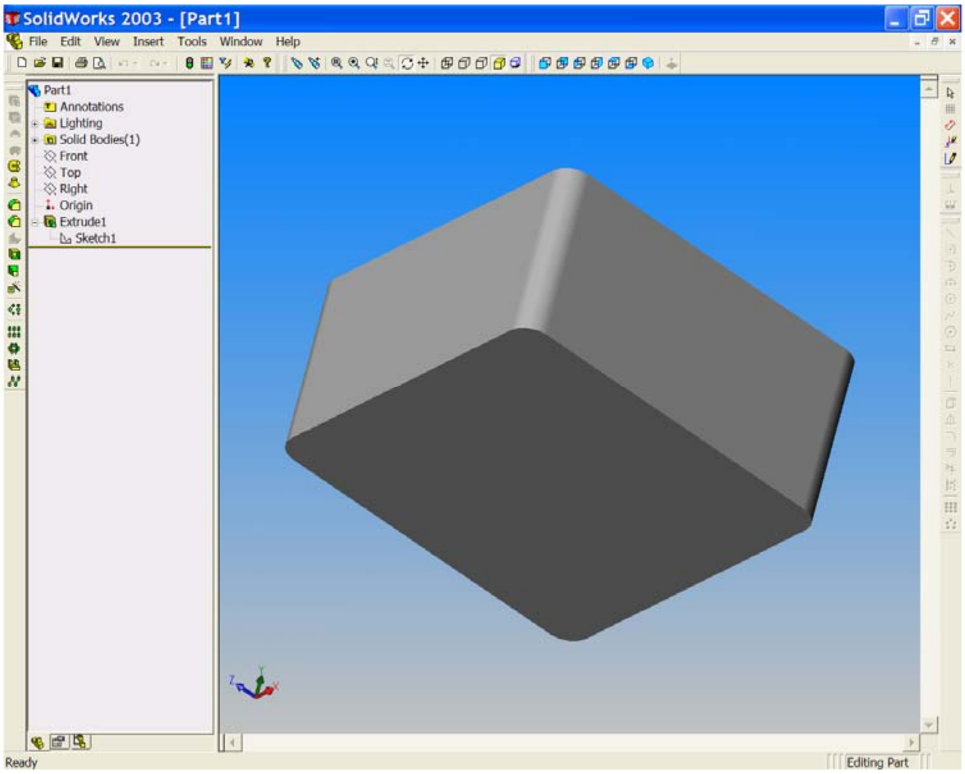


Figure 8

22. Select the bottom face of the model by placing the cursor over the face. Notice the flag appearing as the edges of the face are highlighted in red. Left click to select the face. The selected face is now green (active).

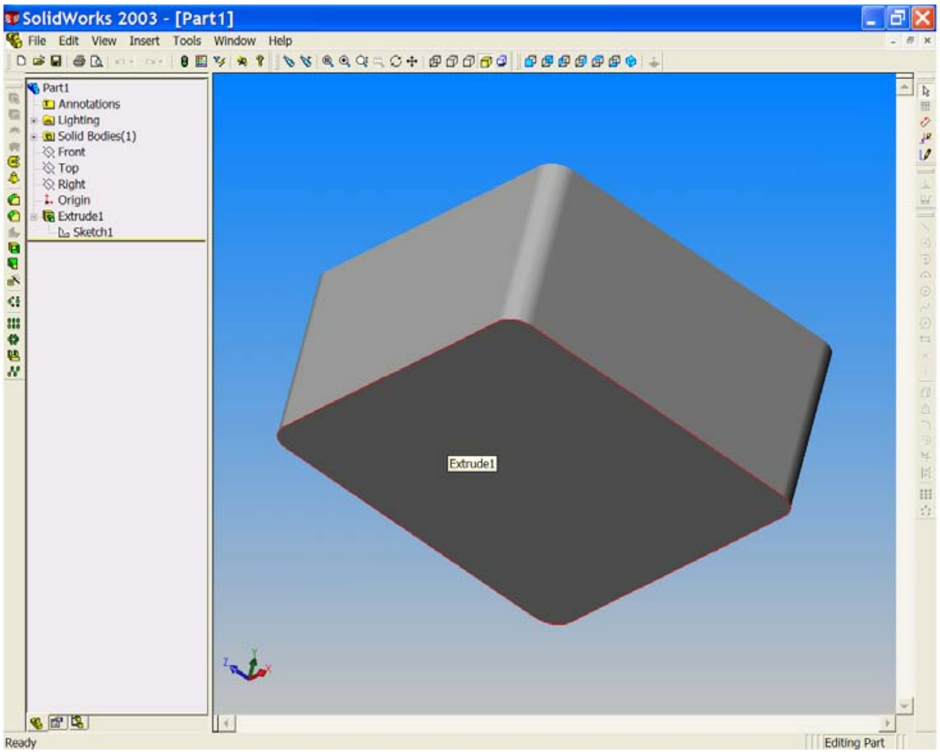


Figure 9

23. Choose '**Shell**' from the Feature menu. The shell property manager opens. Face<1> is already shown in the Faces to remove box and a default value of 10mm is indicated in the thickness window. This value is satisfactory for our model so there is no need to change it. Press the green tick for OK .

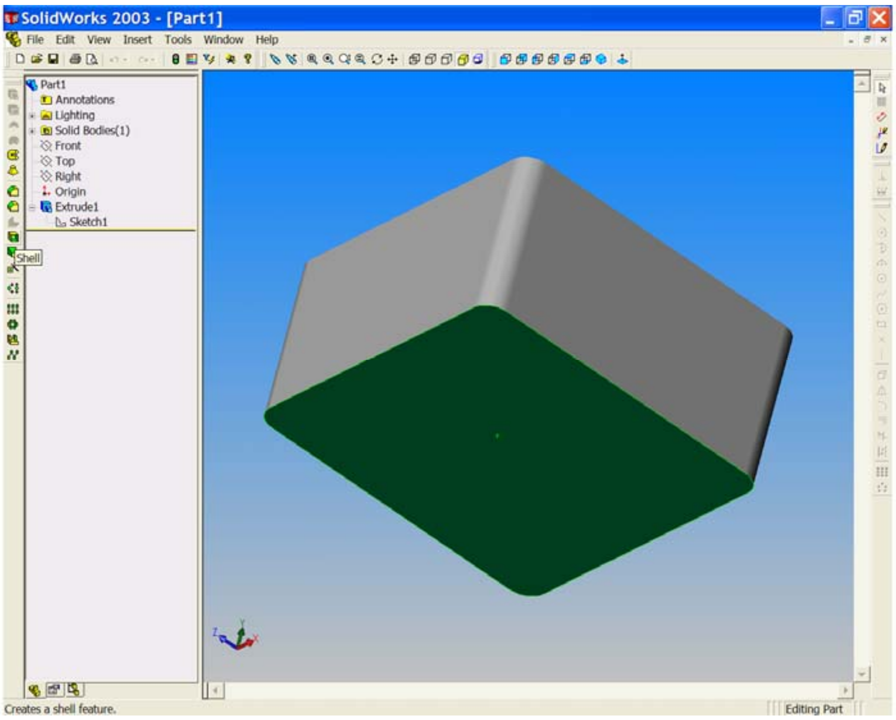


Figure 10

Notice that the base is now shelled or hollowed out with a thickness of 10mm. The Feature Manager Design tree indicates Shell 1.

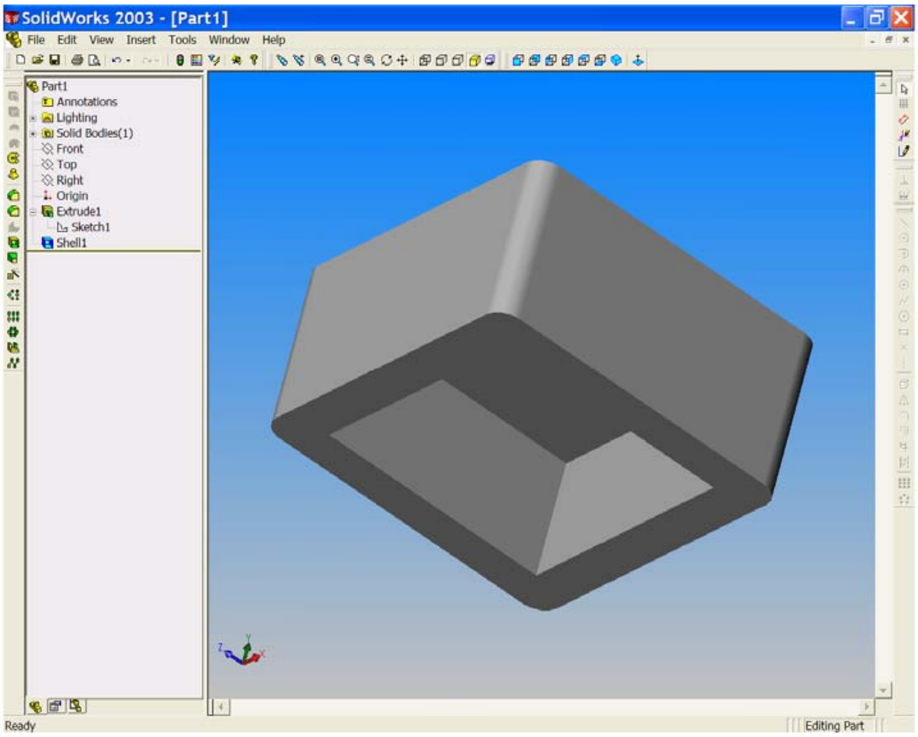


Figure 11

24. From the File menu select '**Save as**'. In the save as dialog box save the model as (give a file name here).

Figure 12

Notice that this new name appears at the top of the screen and at the top of the Feature Manager design tree.

25. Go to **Isometric** view and then select the top surface of the model and then choose '**Normal to**' from the View menu.

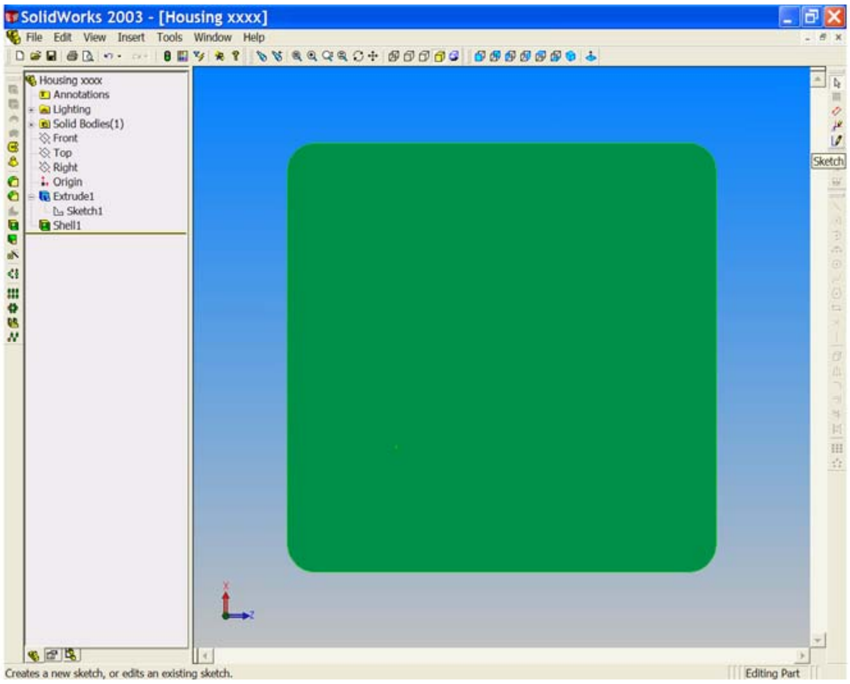


Figure 13

26. Open a new sketch on this surface. Select '**Sketch**' .

27. Choose '**Circle**' from the Sketch tools toolbar.

Notice that the pencil symbol is now accompanied by a circle symbol.

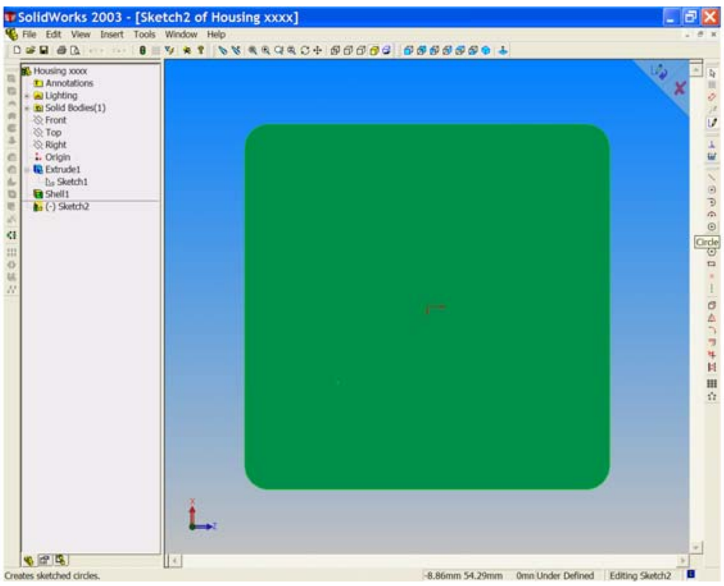


Figure 14

28. Place the pencil symbol on the model origin.

Notice that it changes colour (it becomes yellow and that the circle symbol is replaced by an orange square).

Left click once again to place the circle centre at the model origin. The circle property manager opens. Drag the mouse from the origin to draw the circle.

Notice that the circle centre is fixed at the origin and that the circumference is attached to the mouse.

The circle radius is shown beside the symbol. For this model the circle radius needs to be about 32mm. When the circle radius is about 32mm left click once, to place the circle and end the command.

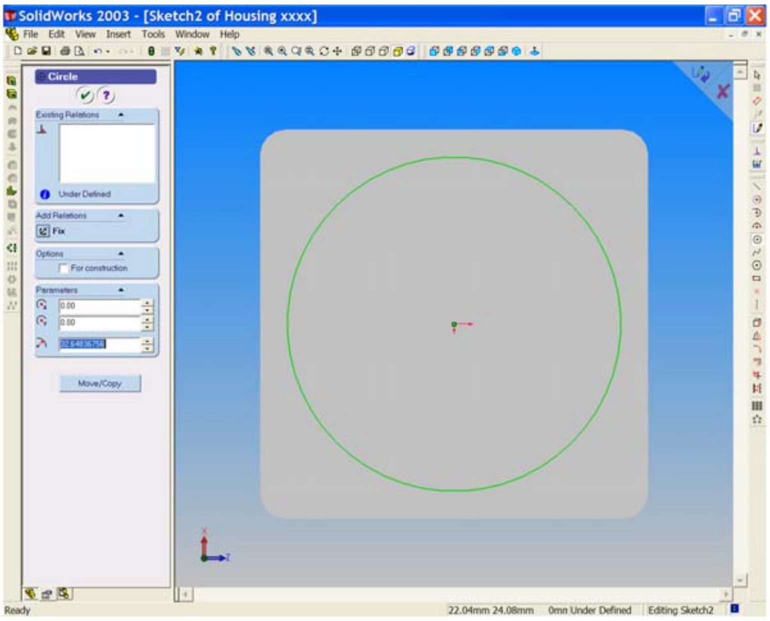


Figure 15

29. Right click once.

By right clicking we can open a shortcut menu to some commands, including the dimension command.

Choose Dimension from he shortcut menu by left clicking on the word '**Dimension**'

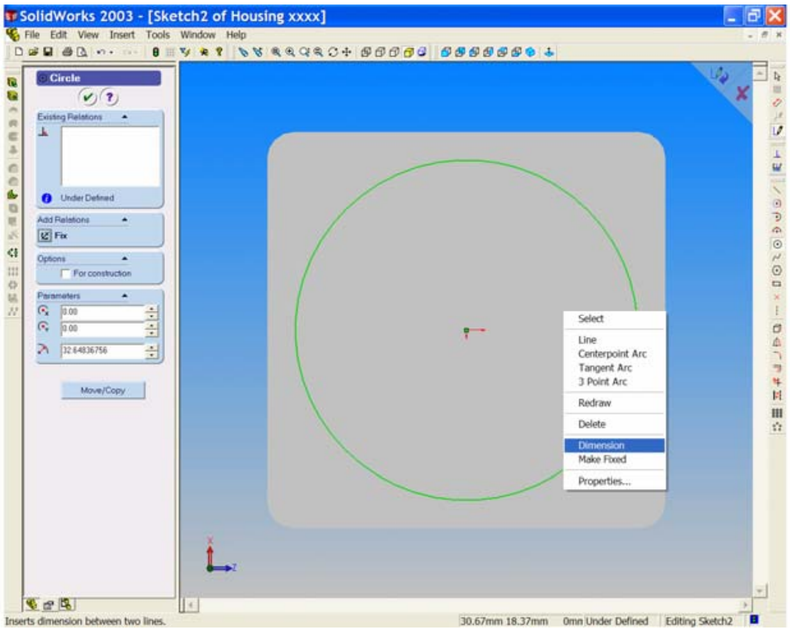


Figure 16

Notice that the circle command is now complete and the circle is blue in colour, fixed to the origin but still under defined. To fully define the circle we must add a dimension.

Select the circle at any point on its circumference and place the dimension at any convenient location. In the modify box delete the value and enter 64mm.

The Ø64 is added to the model and the circle re-sizes to the correct dimension. The circle sketch is now black and fully defined.

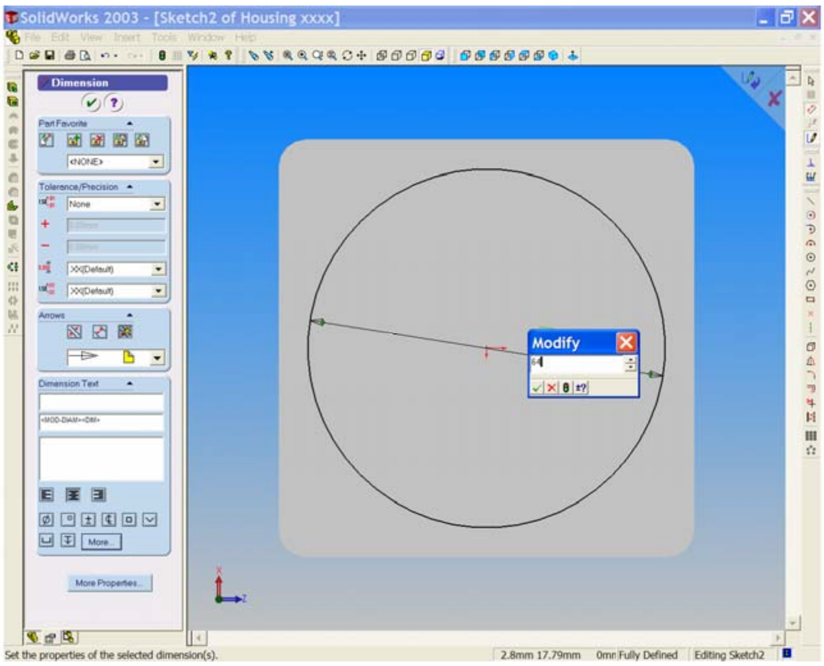


Figure 17

Right click once again and choose select to end the dimension command.

Sketch 2 is added to the Feature Manager Design tree.

30. Select '**Extruded Boss/Base**' from the Features Toolbar. The extrude feature property manager opens with a value of 10mm. Choose '**Isometric**' (from the Standard views menu) to see the model more clearly. The height of the boss should be 38mm, and we wish to merge the result. Enter the value 38 into the depth box and check the merge result box if needed.

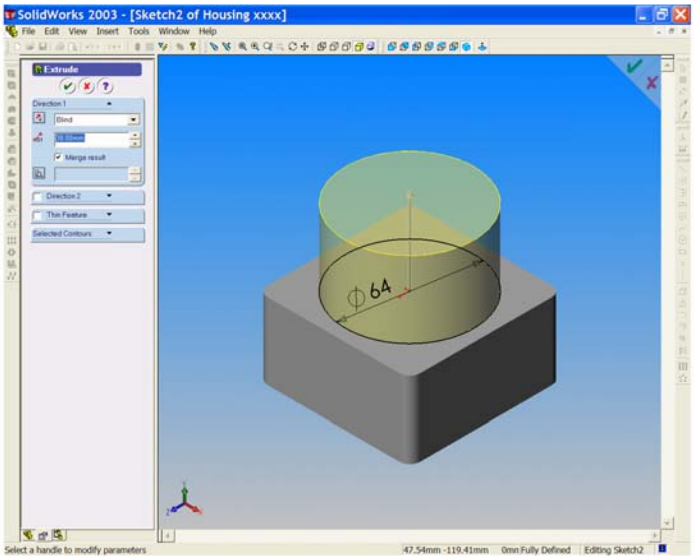


Figure 18

Notice that the circle sketch is extruded from the top surface of the base for a distance of 38mm in the direction of the arrow.

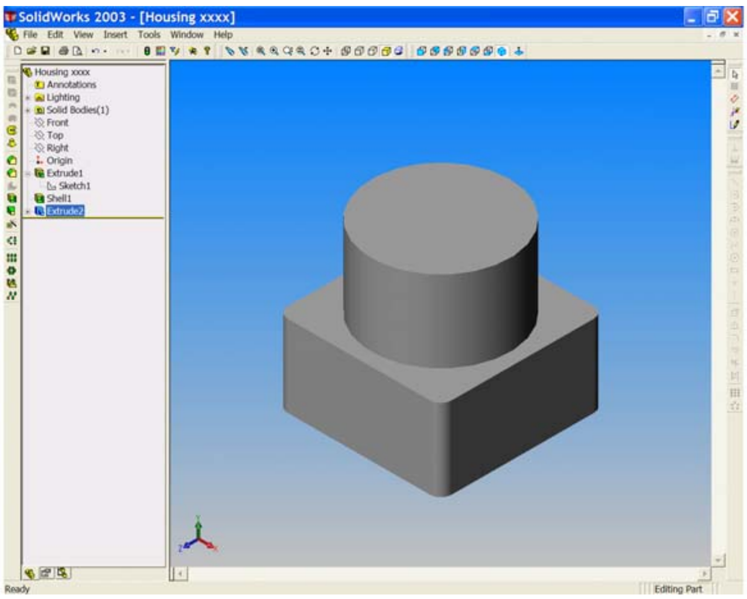


Figure 19

Left click the green tick to accept the extrusion.

Notice that Extrusion 2 is now added to the design tree. Open extrusion 2 as before and note that its base is sketch 2 (the circle).

31. Place the cursor on the top surface of the extruded boss. Notice that the surface is outlined in red and that a flag symbol appears with the cursor arrow. Left click to accept this surface; it turns green to indicate that it now active. Open a third sketch on this surface.

32. Choose '**Circle**' as before. Place the pencil symbol anywhere on the green surface **BUT NOT ON THE ORIGIN**. Draw a circle of any size.

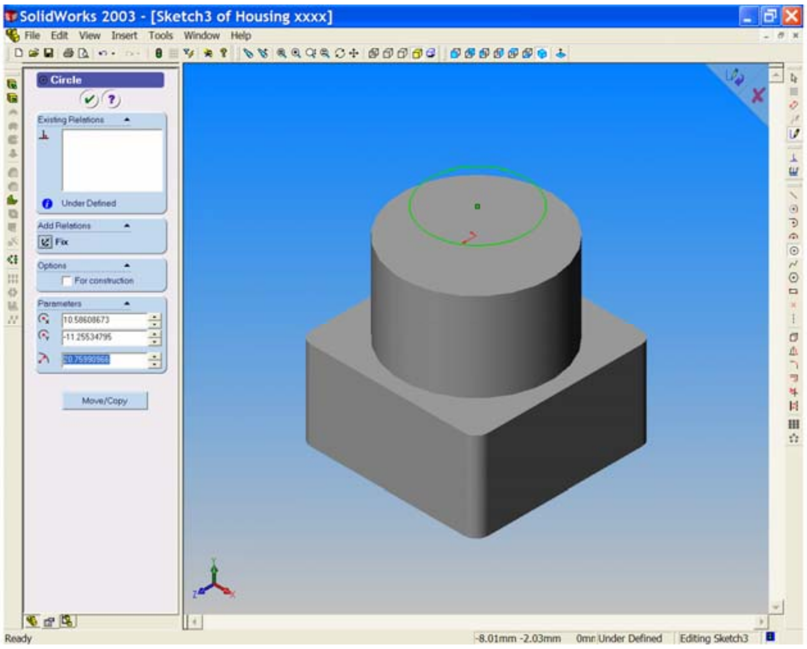


Figure 20

We now need to define this sketched circle.

33. Right click and choose '**Dimension**'. Dimension the circle to be Ø48mm.

To fully define sketch 3 we must now fix its centre. We can do this by using geometry.

34. From the 'Sketch relations' toolbar choose '**Add Relation**'. The Add Relation Property Manager opens.

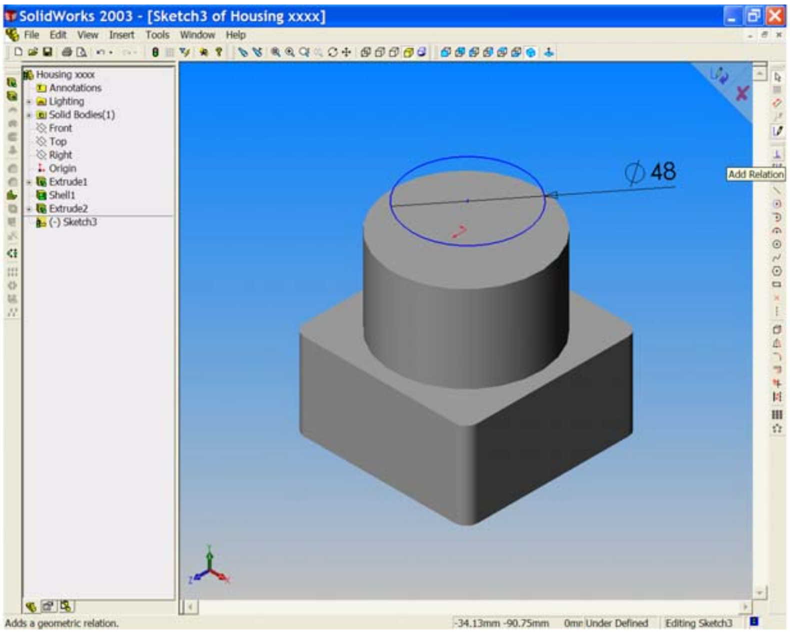


Figure 21

The selected entities box is pink indicating that it is active. Place the cursor on the blue circle and left click to select. The circle is added to the pink box as Arc 1 and the existing relation, its diameter, is listed in the white box.

Place the cursor on the top edge of the Boss. It is now red; left click to select this edge. The edge of the boss is added to the selected entities box (the pink box) as edge <1>.

The geometry relations that are now possible between these two features are now listed under 'Add relations'. We wish to make these two features concentric, i.e. to have the same centre.

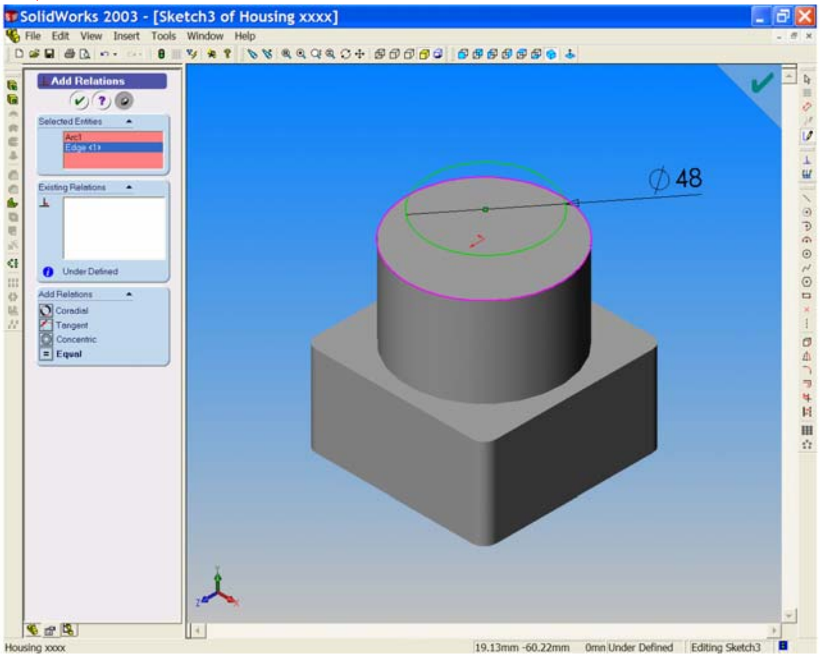


Figure 22

Choose '**Concentric**' by left clicking the mouse. Concentric 0 is added to the existing relations box and the green circle (sketch 3) moves so that its centre coincides with the boss centre. Choose the green tick to end the command.

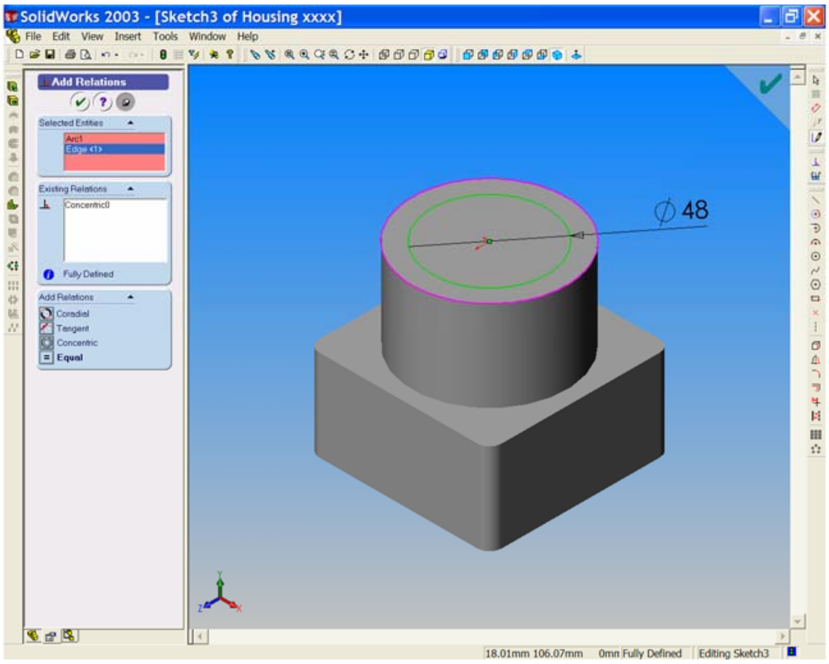


Figure 23

35. We now wish to cut a counterbore hole in the top of the Boss.

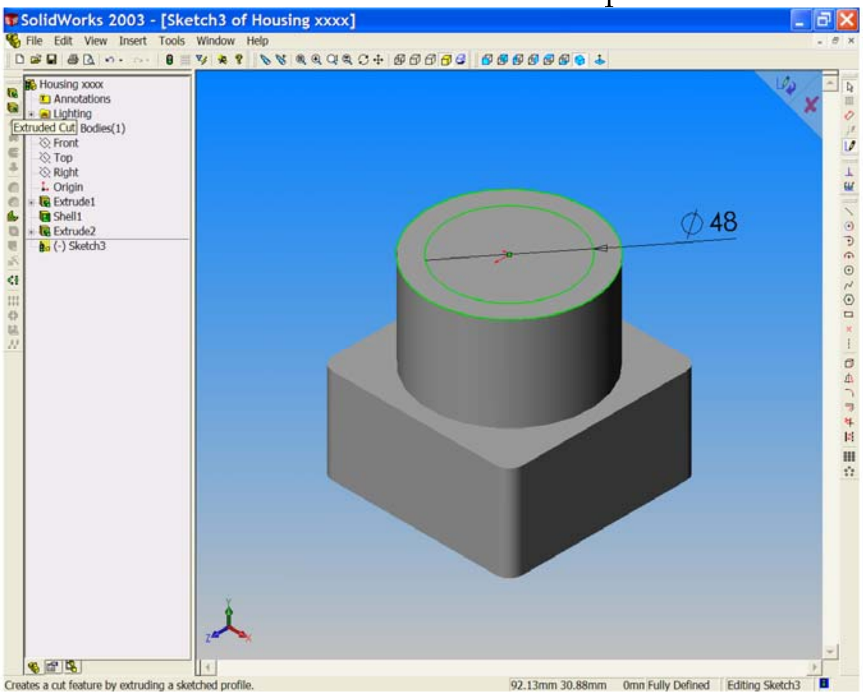


Figure 24

From the Features toolbar, select '**Extruded Cut**'. The Extruded cut Property Manager opens. In the depth window change the value to

12mm. Check that the end condition window shows 'Blind'. Select OK to cut the counterbore hole in the top pf the Boss.

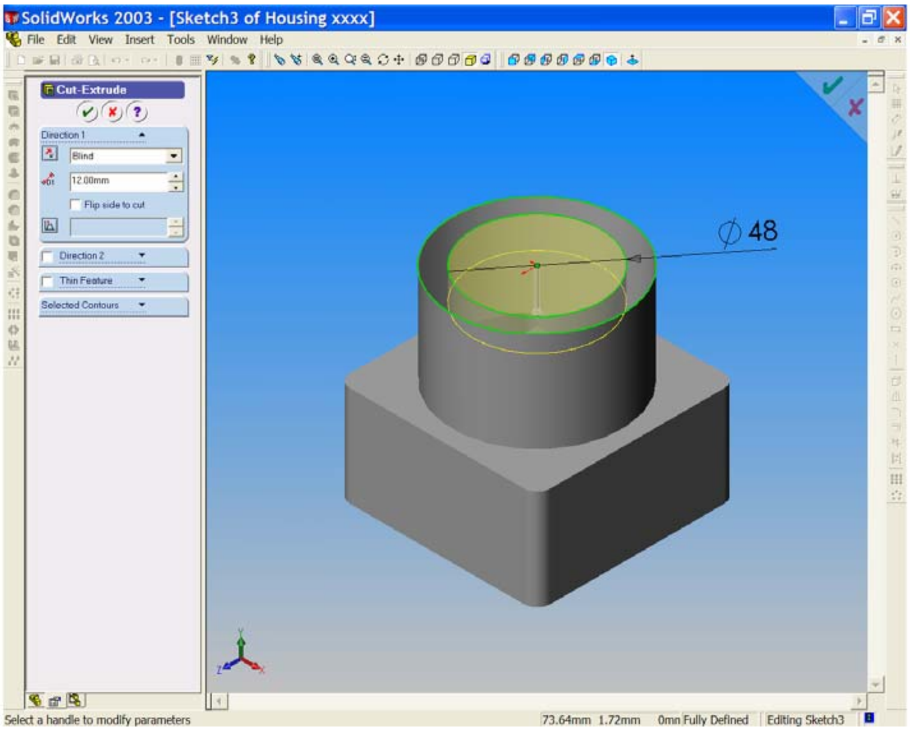


Figure 25

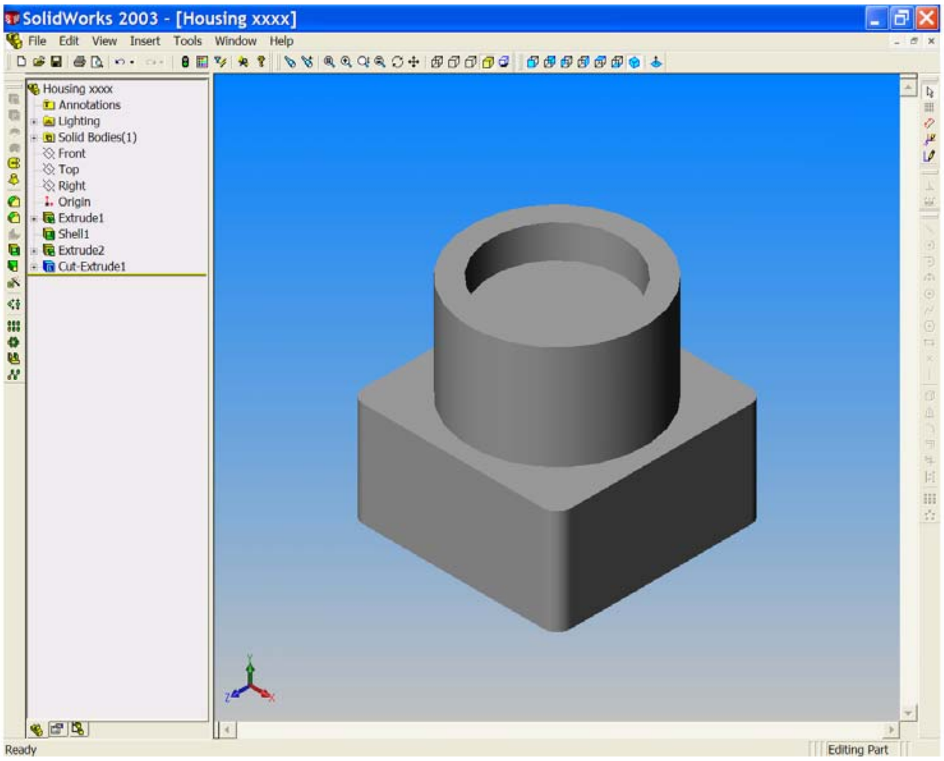


Figure 26

36. Save!!! (Regularly)

We now wish to cut a hole from the bottom of the counterbore all through the boss and into the shelled base.

37. Select the bottom of the counterbore hole. Open a new sketch on this surface and draw a Ø30mm circle concentric with the boss.

38. With sketch 4 active choose '**Extruded Cut**'. This time select the small down arrow at the right side of the '**End Condition**' window. The option to choose is '**Through All**'.

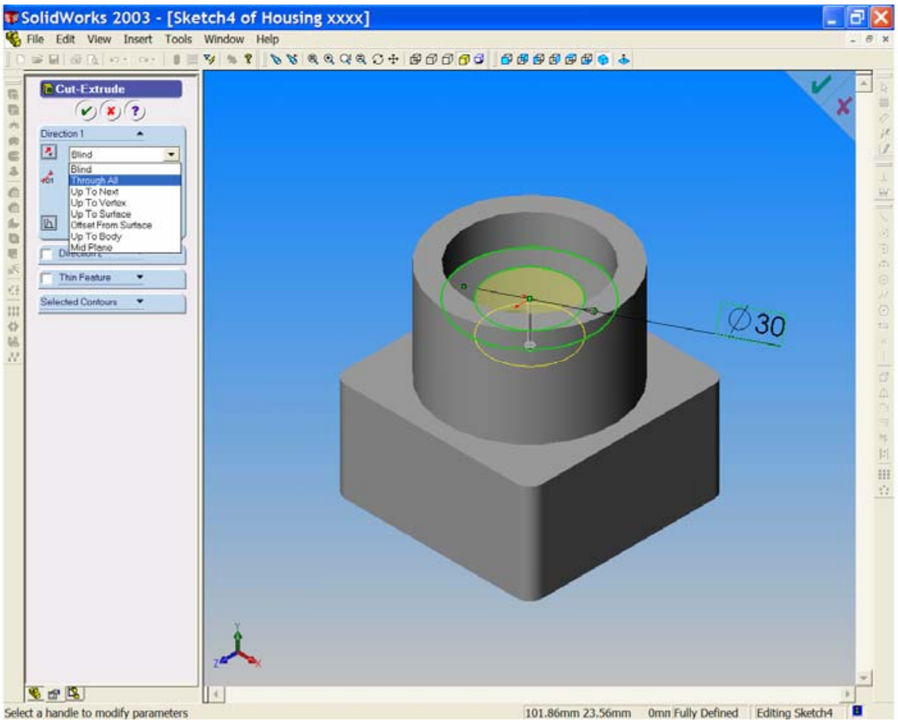


Figure 27

Notice that we do not need to give a depth when using this option.

Select OK to cut the hole.

Extrude-Cut 2 is added to the design tree. Use the rotate View button to look at the model from any direction. Return to the isometric view when you are satisfied that the hole is cut through.

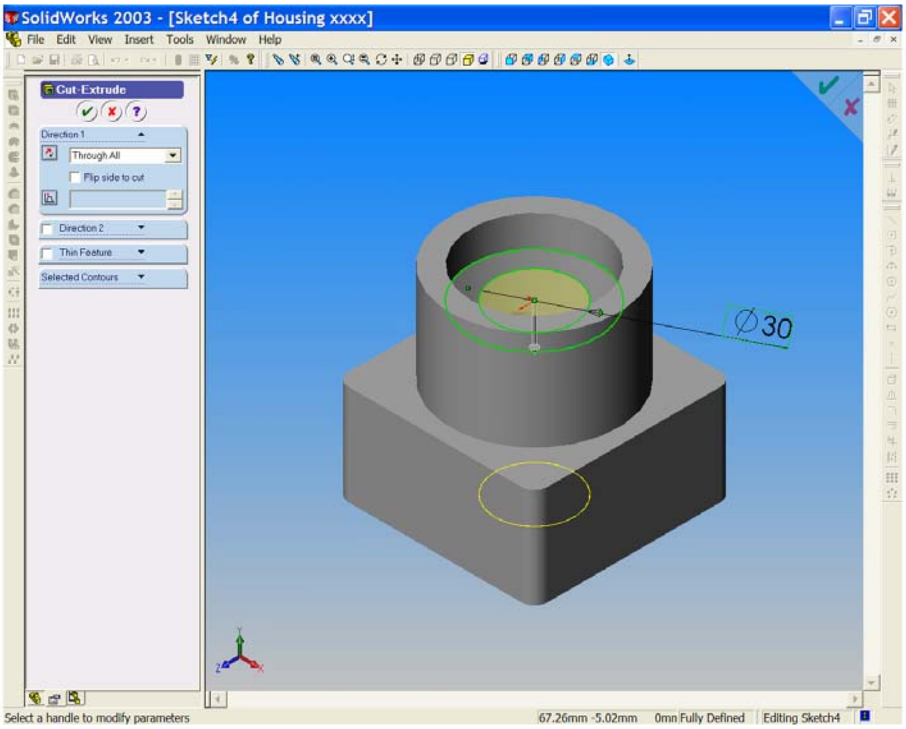


Figure 28

39. *We need to add a fillet to the bottom of the boss and the top surface of the square base. We can achieve this with the fillet command from the Feature Toolbar.*

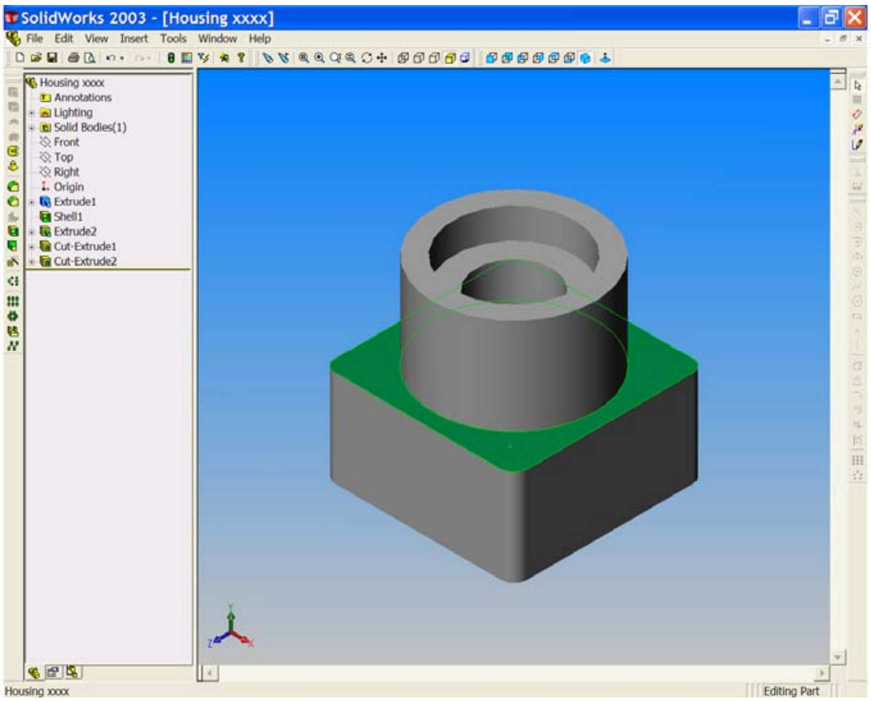


Figure 29

Select the top surface of the square base. It is now outlined and shaded green. Choose '**Fillet**' from the feature toolbar. Face <1> is already selected and shown in the '**Items to fillet**' box (the pink box). Change the fillet radius to 3mm.

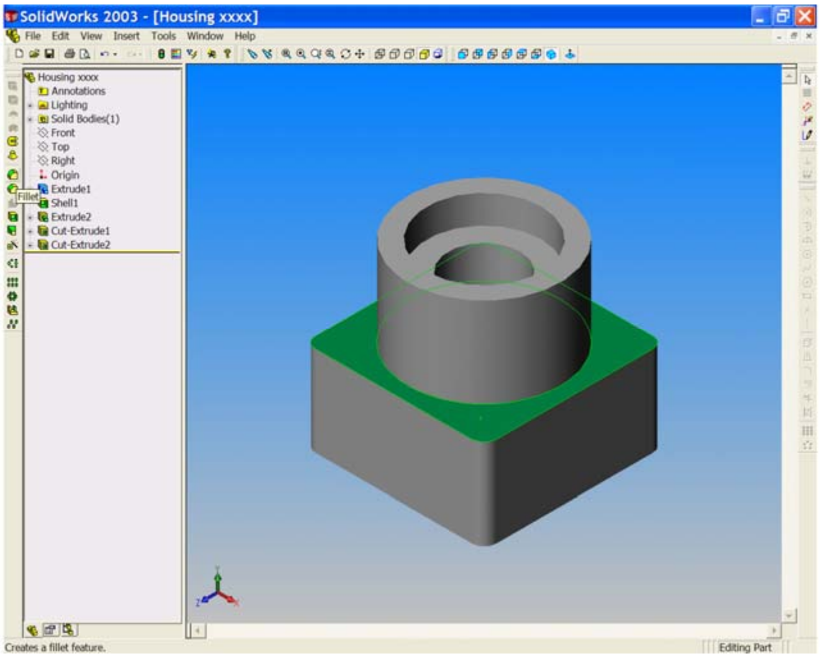


Figure 30

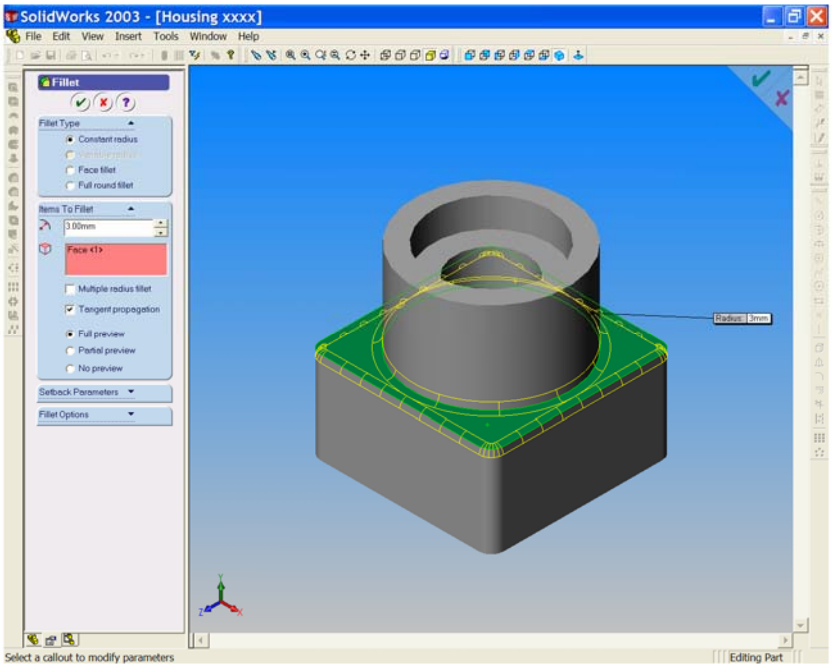


Figure 31

Notice the preview of the fillet appearing on the model, it will be shown in yellow.

Choose OK to accept the fillet.

40. Save your work !!!!

At this stage of the model construction we may add fillets to the hidden edges inside the square base.

41. From the view toolbar select '**Hidden lines visible**'. The sharp edges that need to be filleted can be seen and are shown as broken lines.

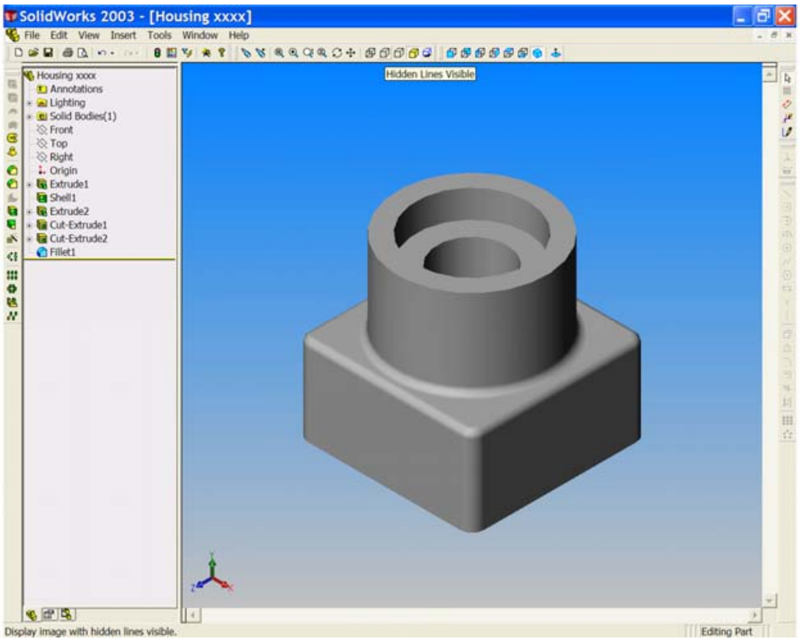


Figure 32

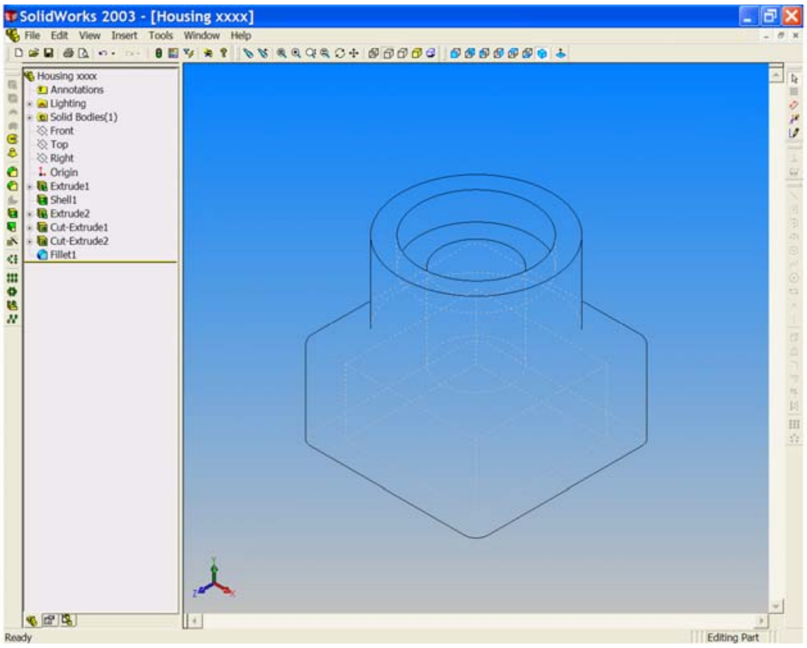


Figure 33

We need to select each of them one by one and to do this we will use the control key on the keyboard.

42. Press the control key on the keyboard and ***HOLD IT DOWN***. Select the four vertical edges. They will become red when your cursor is on them. As you select each one it will turn green.

Still holding the control key down, select the four horizontal edges in the same way. When all eight edges have been selected, release the control key.

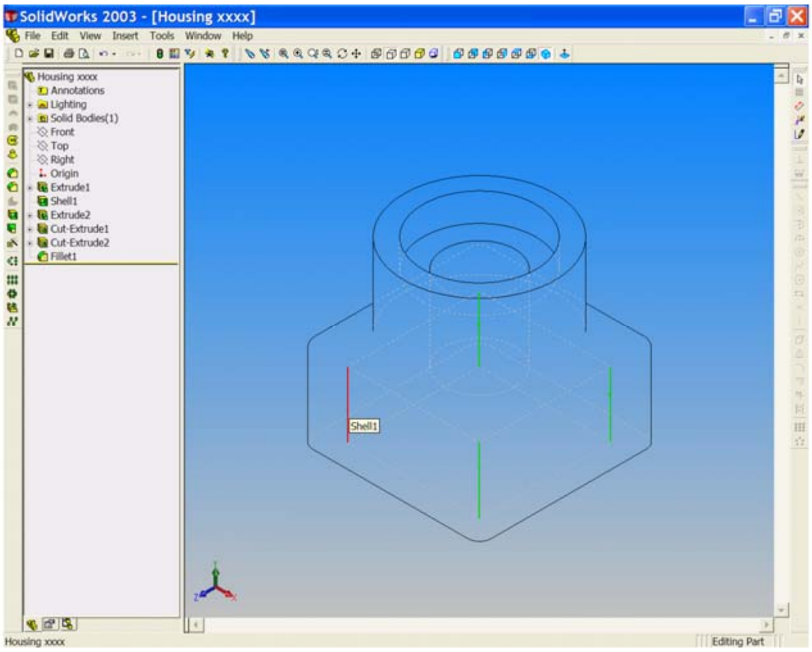


Figure 34

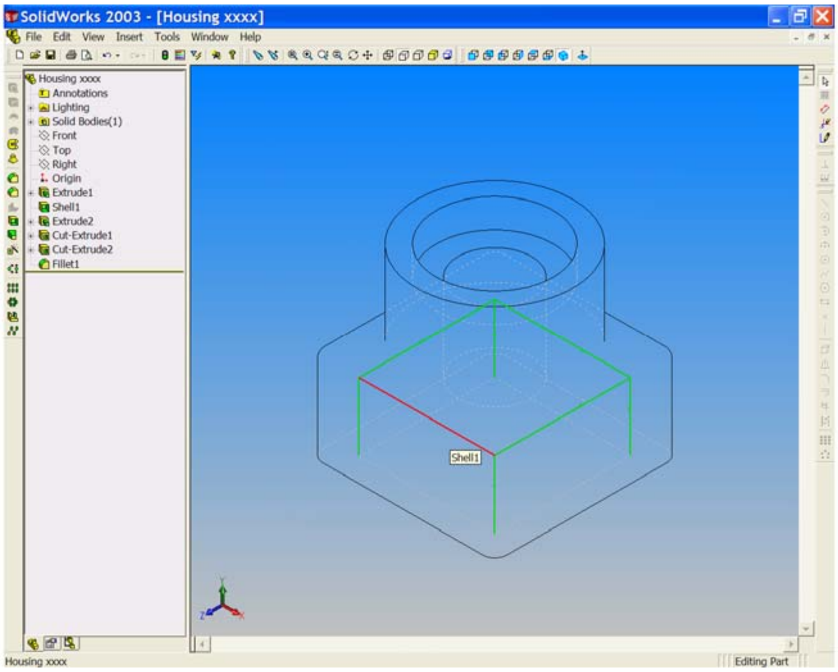


Figure 35

43. Choose the '**Fillet**' button from the Feature toolbar. Since the 3mm setting from the last command should still be active in the Fillet Property Manager, your model should show a yellow preview of the internal fillets.

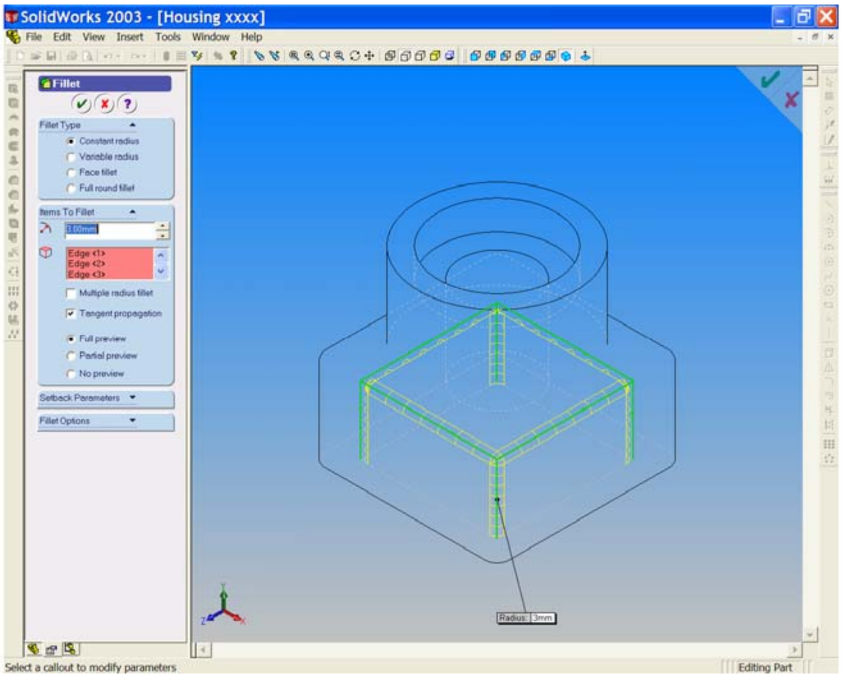


Figure 36

44. Choose **OK** to accept the fillets.

45. Sketch '**Shaded**' from the View toolbar and rotate the model to see your internal fillets. Use the '**Isometric**' button to return to the standard isometric view.

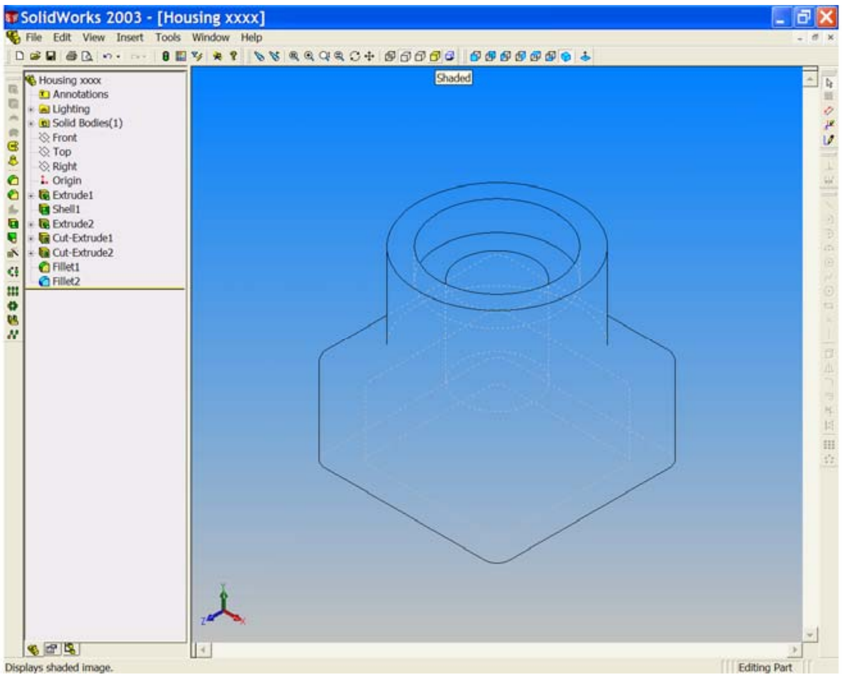


Figure 37

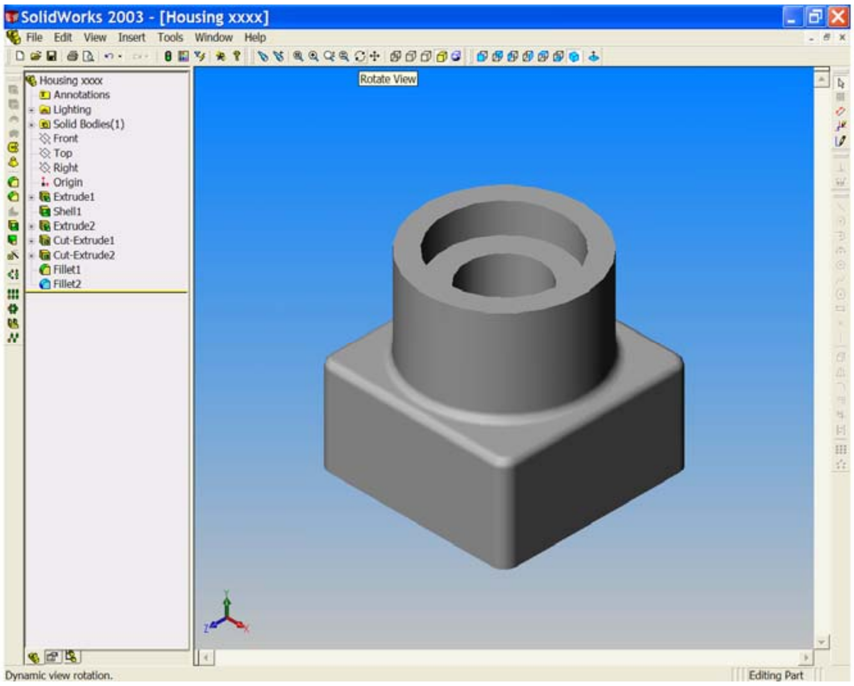


Figure 38

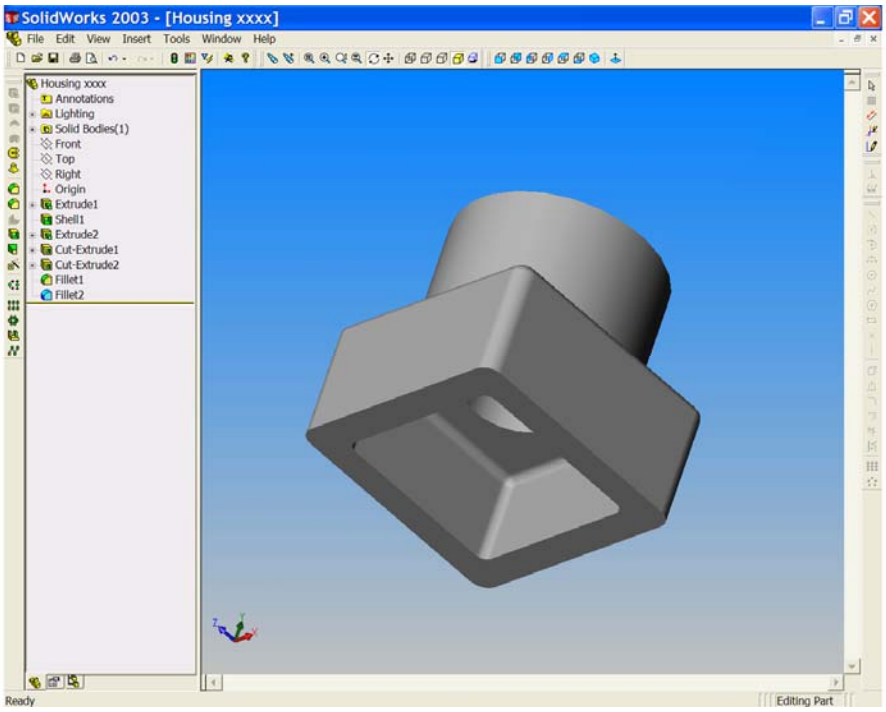


Figure 39

Save your work!

We need to add boss to the right face of the square base.

46. Select the right face of the square base, look for the flag and notice the selected surface colour change as before.

47. Choose '**Normal to**'  and open a sketch  on this surface.

48. Select ‘**Circle**’  and draw a circle towards the right side of the surface.

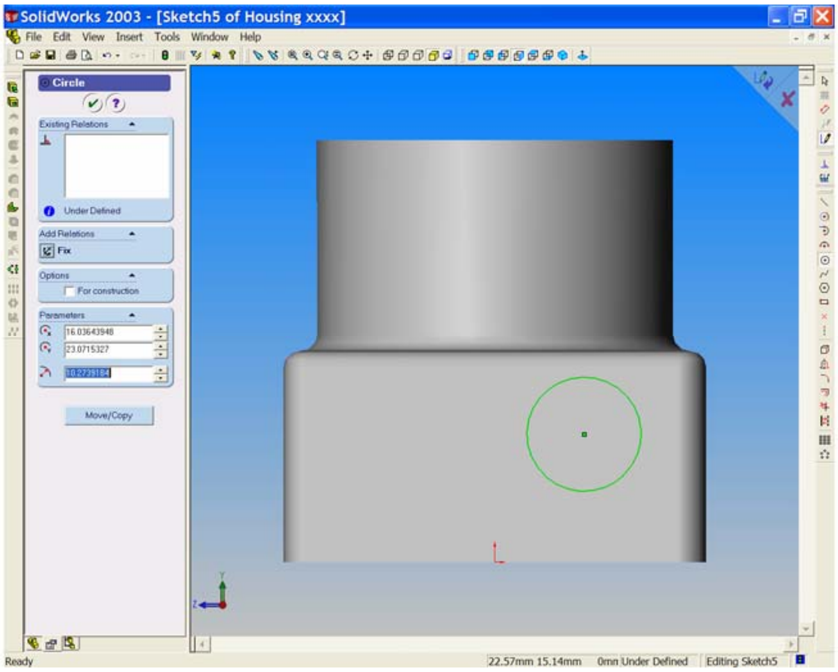


Figure 40

49. Right click and choose ‘**Dimension**’ from the short-cut menu. Dimension the circle to have a Ø24mm.

50. Continue with the dimension tool to position the circle centre at 24mm from the right edge and 24mm from the top surface of the square base. Sketch 5 is now fully defined.

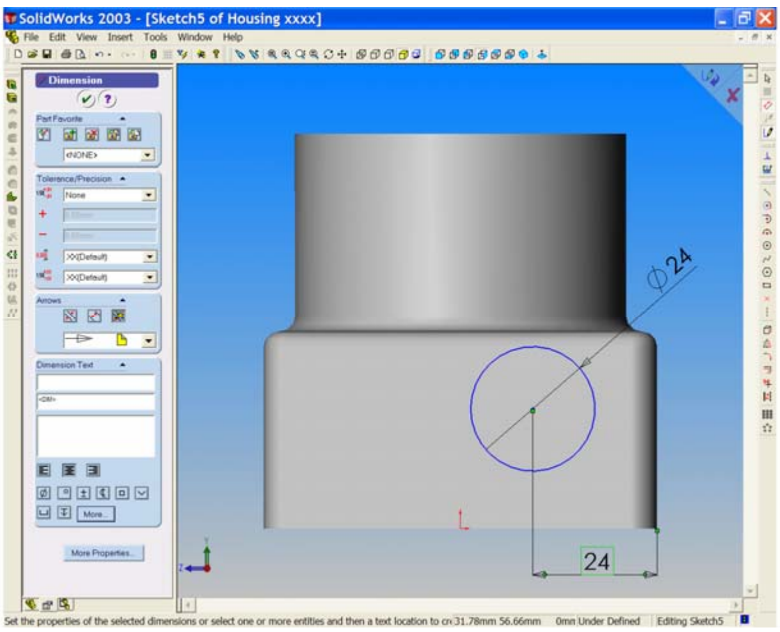


Figure 41

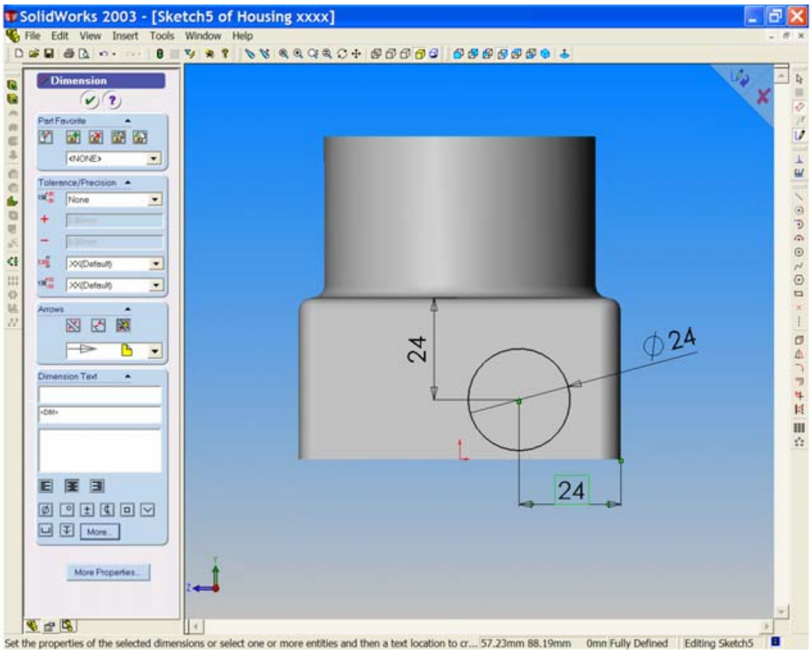


Figure 42

51. Use the '**Extrude Boss/Base**' tool to extrude the circle for a distance of 10mm. Use the isometric button to help you see what you are doing.

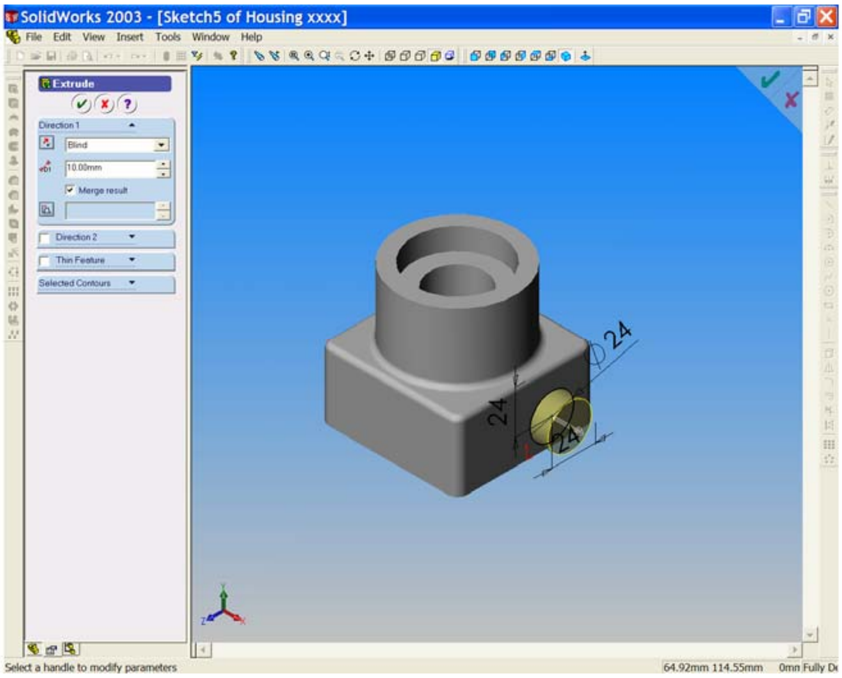


Figure 43

52. Add a 3mm fillet to the new boss feature. Select the circle at the base of the new boss and then choose the '**Fillet**' button from the features toolbar. Enter a value of 3mm into the radius window and then **OK**.

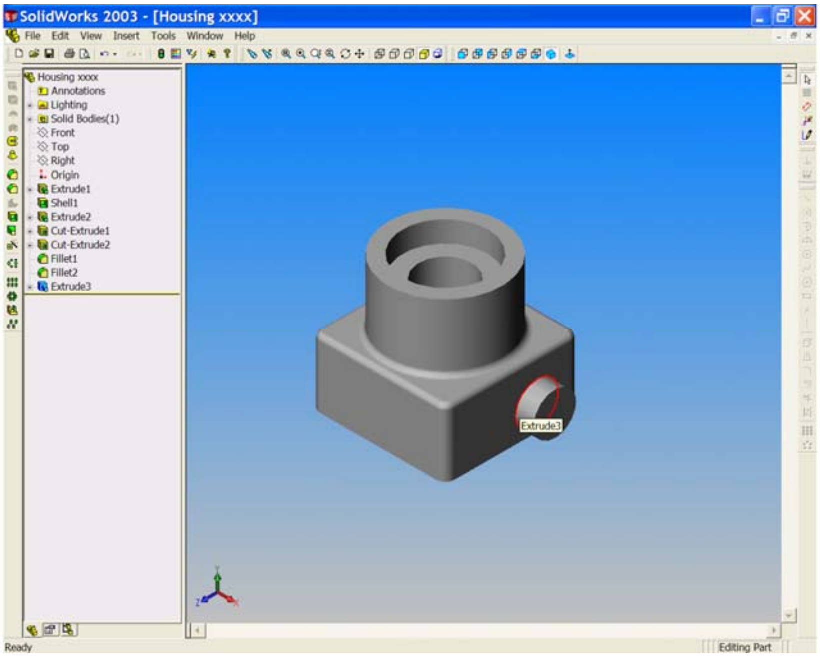


Figure 44

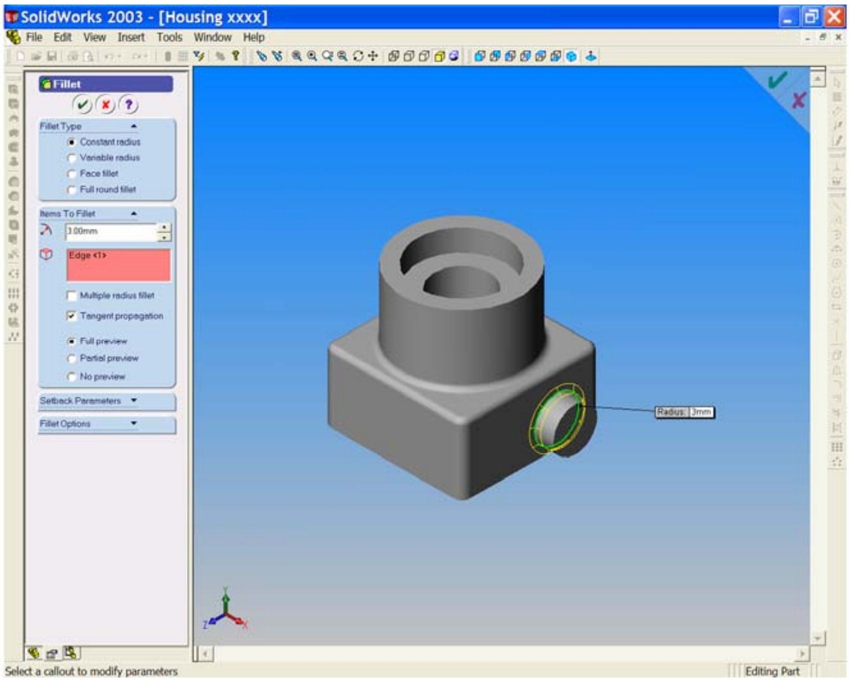


Figure 45

53. The last requirement for this model is a hole through this boss. Select the flat outer surface of the boss and open a '**Sketch**'. Choose '**Circle**' and draw a circle on this surface. Right click and dimension the circle to have a Ø16mm.

54. Now add a 'Concentric relation' to place the circle. Choose '**Add relation**', select the circle and then select the edge of the boss. Choose 'concentric' from the Add relations Property Manager.

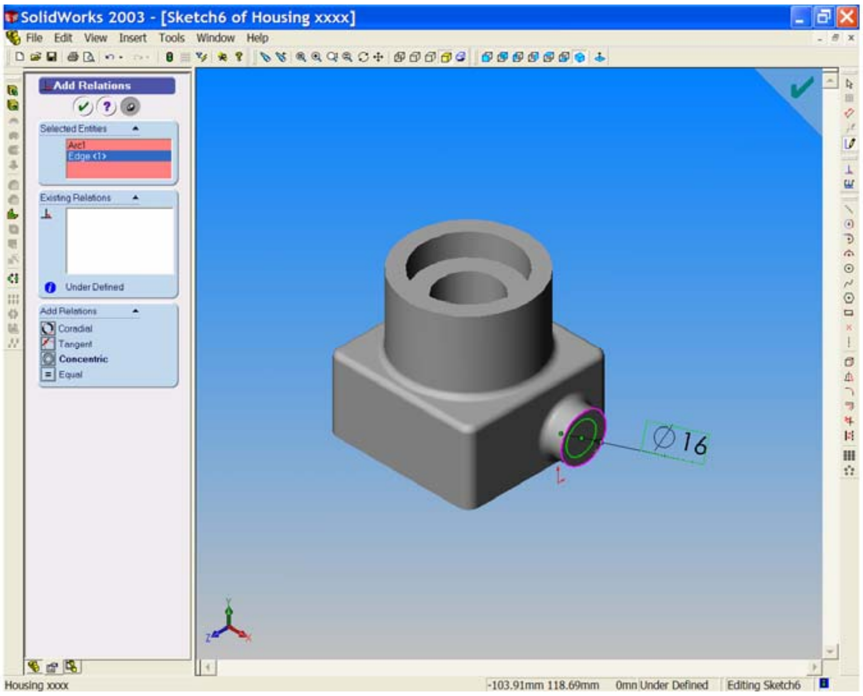


Figure 46

55. Use '**Extruded Cut**' to make the hole. Be sure to keep the end condition as '**Blind**'. Change the depth to 20mm and click **OK**.

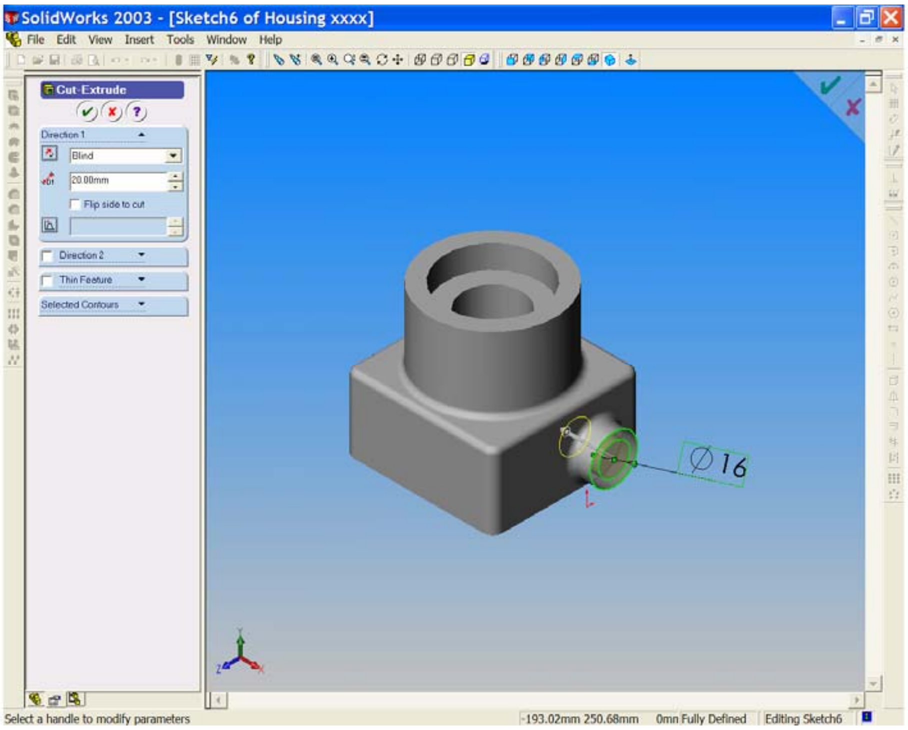


Figure 47

Rotate the model to see inside the base.

56. To change the colour of the model, first select the file name at the top of the feature manager design tree.

It is now highlighted, select '**Edit Colour**' from the Standard toolbar. You may now choose any colour you like! Try one of them or define your own colour. When you have selected a colour pick the 'Apply' button and then **OK**. Your model changes to your chosen colour!

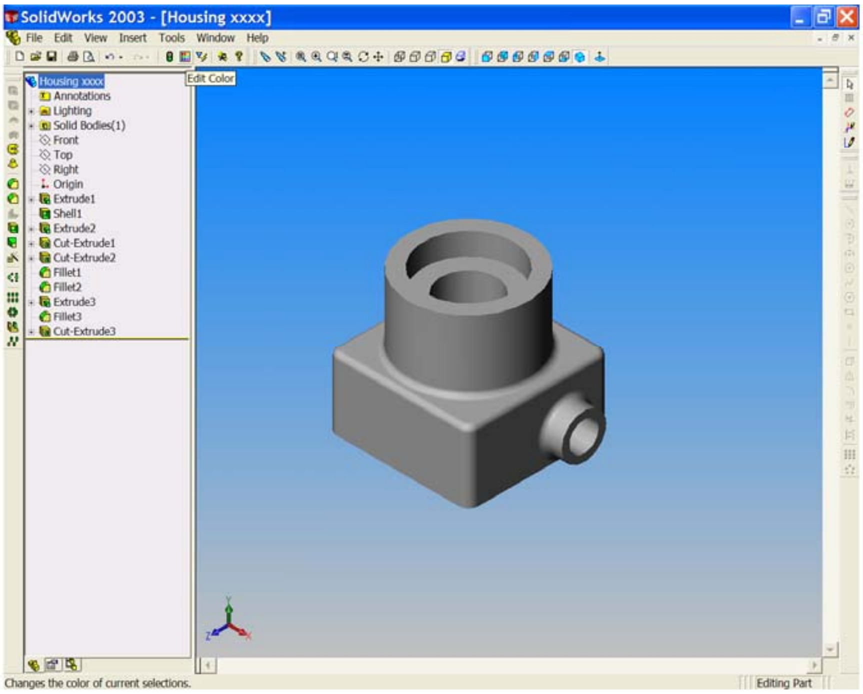


Figure 48

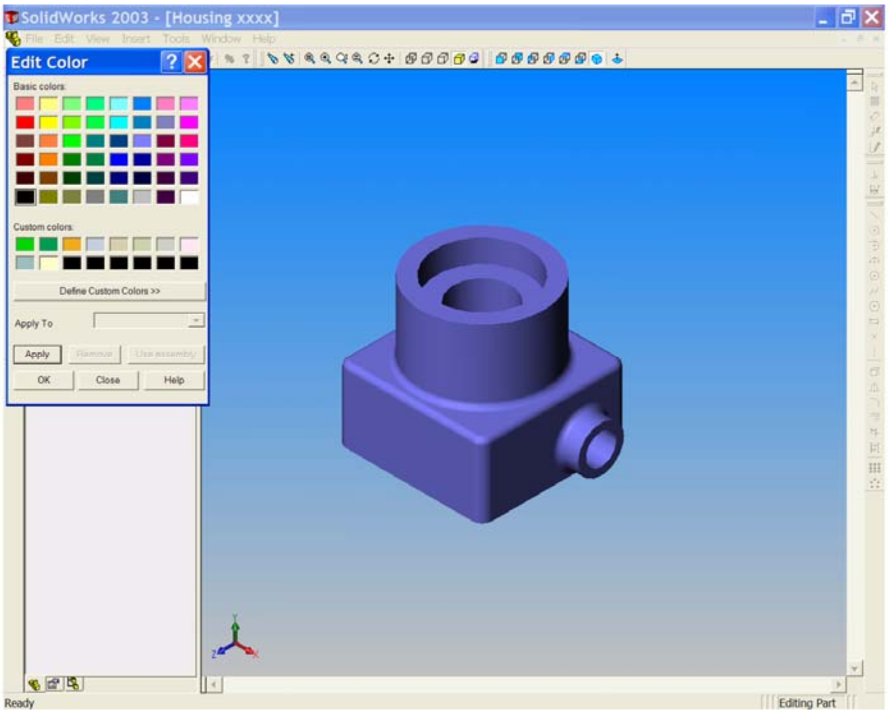


Figure 49

*Your model is complete so ………. **SAVE YOUR WORK** !*

Module 5: To create the 2-D Drawing from a 3-D Model

1. Start the program.

2. Select '**Open Document**' .

3. Choose the **Housing Exercise** from the location where you have it saved.

   i. Note the 'preview' window, this window permits you to look at the file before opening it.
   ii. The 'Files of type' window. The down arrow allows you to display only your selected type of file from the drop down menu. In this case the type of file is a 'part file' and has the file extension (.prt) or (*.sldprt).*

   Select '**open**'.

4. With the Housing file open select '**New**' and choose '**Draw**' Draw from the tutorial window. See Figure 1.

   Notice that a preview of a standard drawing border is shown.

   Select '**open**'.

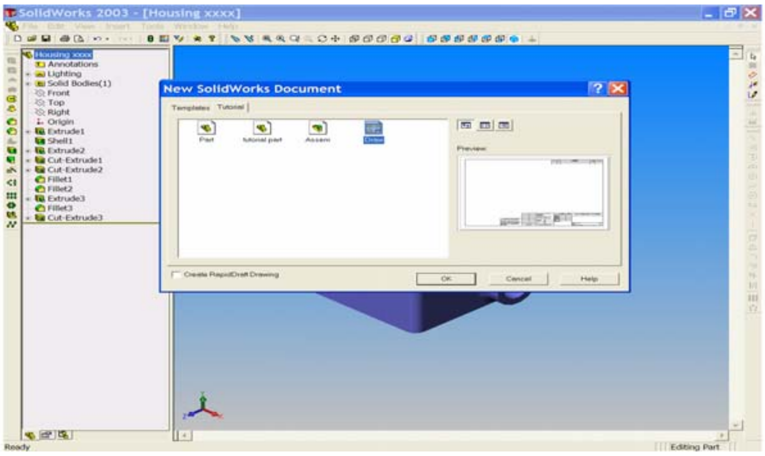


Figure 1

5. In the white drawing area '**Right click**' once to open a short-cut window. Select '**Properties**' to open the '**Sheet Setup**' box. See Figure 2. In the sheet set-up box use the down arrow at the right side of the paper size box to choose the paper size, in this case, **A3-Landscape**.

In the Sheet Format window select **A3-Landscape** to match the paper size. Leave the scale at 1:2 for the time being. Ensure that the type of projection selected is '**Third angle**'. Click **OK** when these selections are made. See Figure 3.

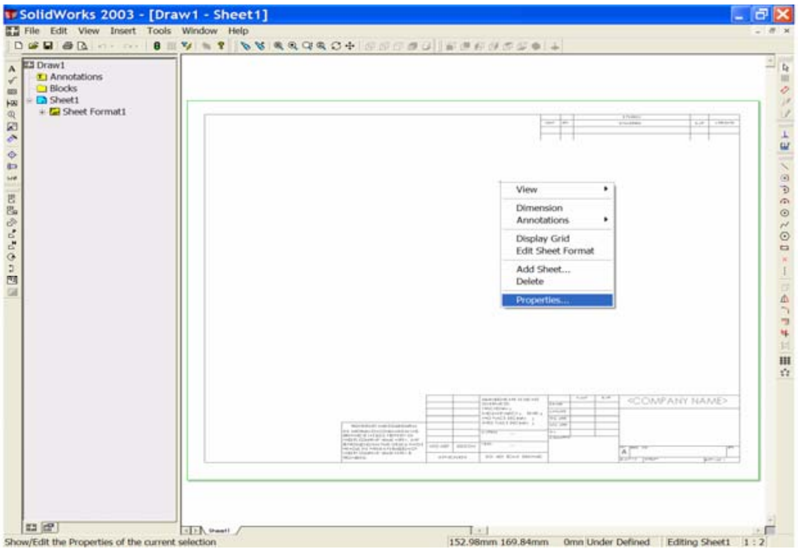


Figure 2

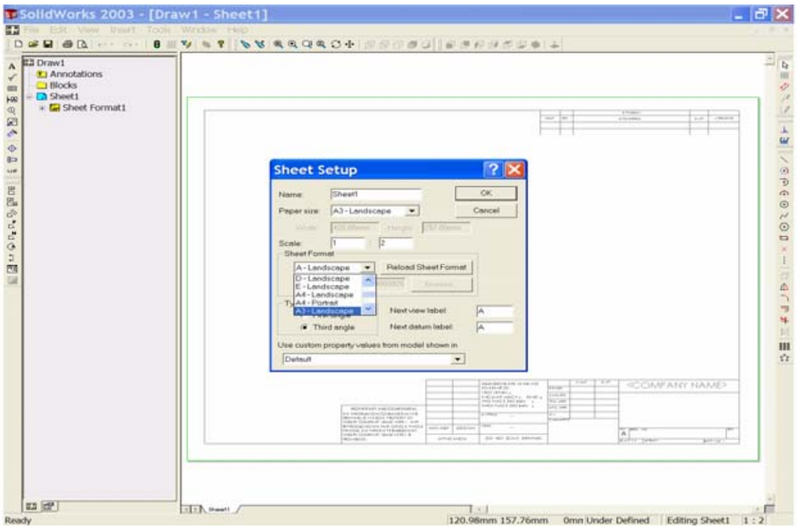


Figure 3

The default border is now replaced by your selected border.

6. Use the '**Zoom to Area**' button from the View Toolbar to examine the border contents. Left click the '**Zoom to Area**' button and draw a window around the '**Title block**' of the border. To do this, place the cursor symbol at the upper left corner of the Title Block, left click and ***HOLD THE LEFT BUTTON DOWN***. Drag the cursor symbol to the lower right corner of the Title Block to draw a green window. Release the mouse button and the contents of the window are scaled to fill the screen. See Figure 4. Select the '**Zoom to Area**' button once again to end the command.

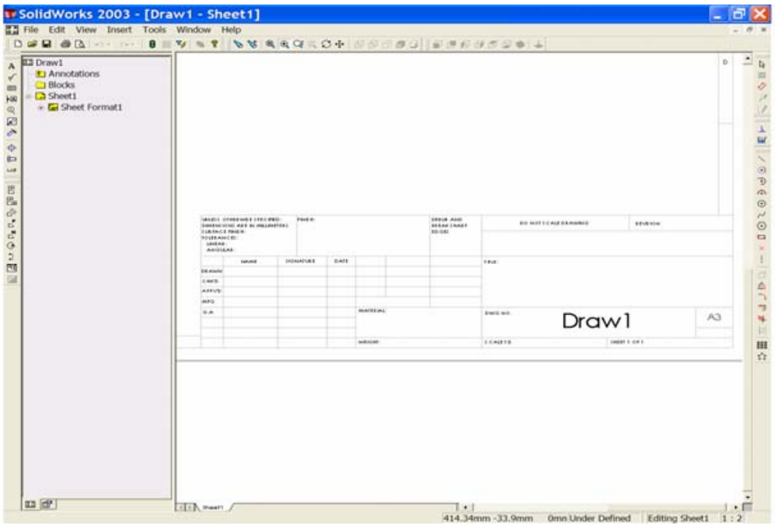


Figure 4

*Notice that there are some new Toolbars available at the left side of the screen. The one at the top should be the '**Annotations Toolbar**' and the one under it should be the '**Drawing Toolbar**'.*

7. From the '**Annotations Toolbar**' choose '**Note**'.

   *The **Note Property Manager** opens and a **NOTE** box is fixed to the cursor.*

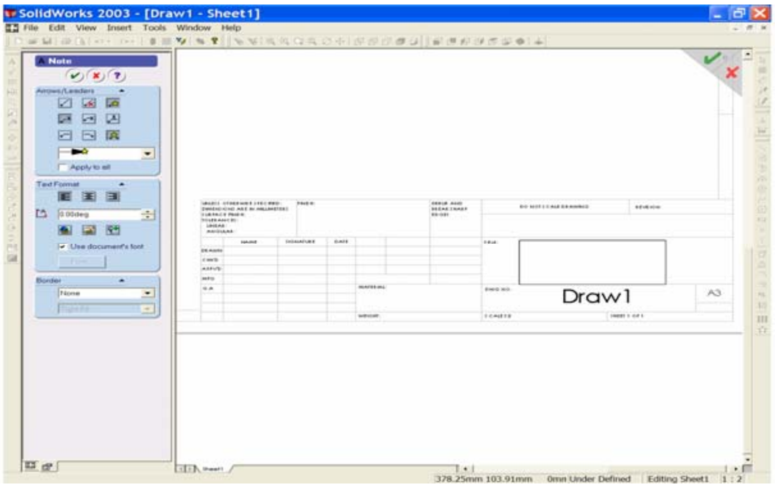


Figure 5

Place the Note box in the Title Area of the Title Block. See Figure 5.

This note box is now active, but the font size is too big.

In the Note Property Manager de-select '**Use document's font**'.

By de-selecting this function we now have access to the font button.

Choose the font button to open the choose font box. In the '**Font**', area scroll to find '**Book Antigua**' and select. In the Height area, select '**Points**' and scroll to select a value of **20**. See Figure 6.

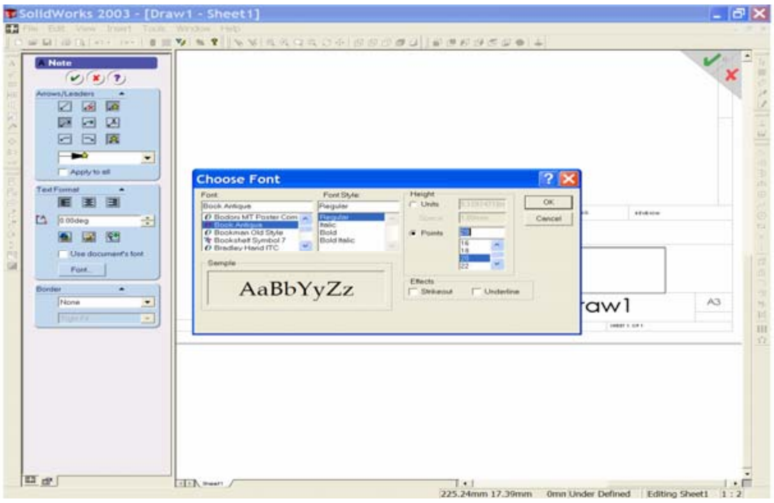


Figure 6

Note that while the units box is disabled (greyed out) its value still changes. Note also that a font sample is shown on screen.

When these changes are made, select **OK**.

8. Left click in the Note box and type the drawing title '**Housing Body**'. When complete, select the green tick in the Note Property Manager.

9. Add '**K.F.U.P.M. Mech. Eng. Dept.**' in the empty box above the title. Adjust the font size to make this new note fit the space available.

10. In the material box add '**Cast Iron**'.

11. Under the Name Box add your '**family name**'. Under signature add '**your ID Number**'. Under Date add the '**date the drawing was made**'. See Figure 7.

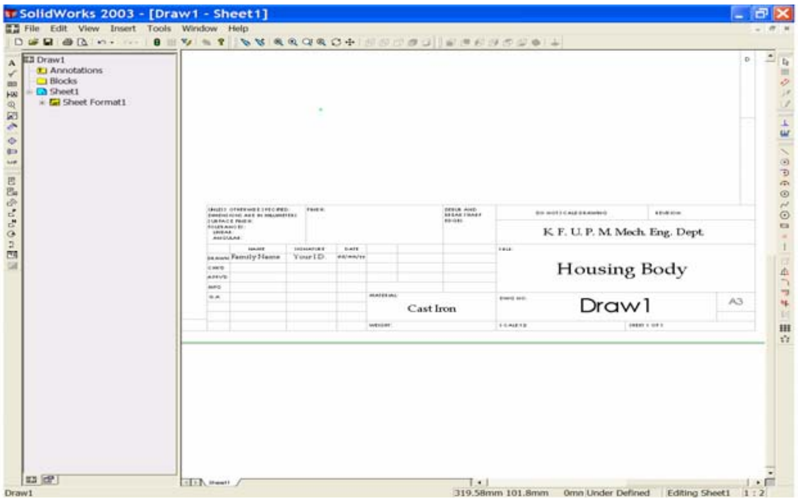


Figure 7

Note that a Dwg No (Drawing Number) is already in place. This drawing number will be the same as your file name.

12. From '**File**' select '**save as'** and save your work. See Figure 8.

Note that the file type extension is now (.drw) or (*.slddrw).*

Use the '**rebuild'** button  to update your drawing sheet.

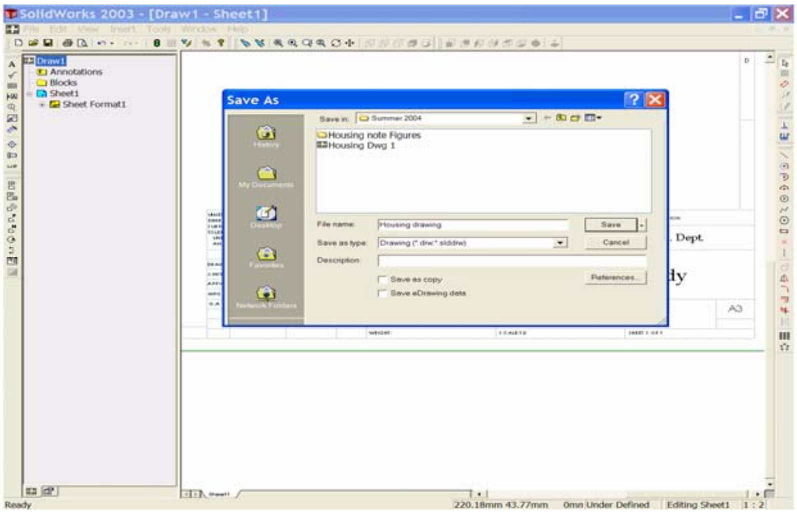


Figure 8

13. To modify the drawing number we must **right click in the drawing area** and select '**Edit Sheet Format**' we can now select **the drawing number**. See Figure 9. **Right click on the drawing name** and select '**Edit text'**. Choose '**Font**' from the Note Property Manager and change the font to match the other notes that you have just added to the sheet. Choose **OK** to update your drawing sheet. See Figure 10.

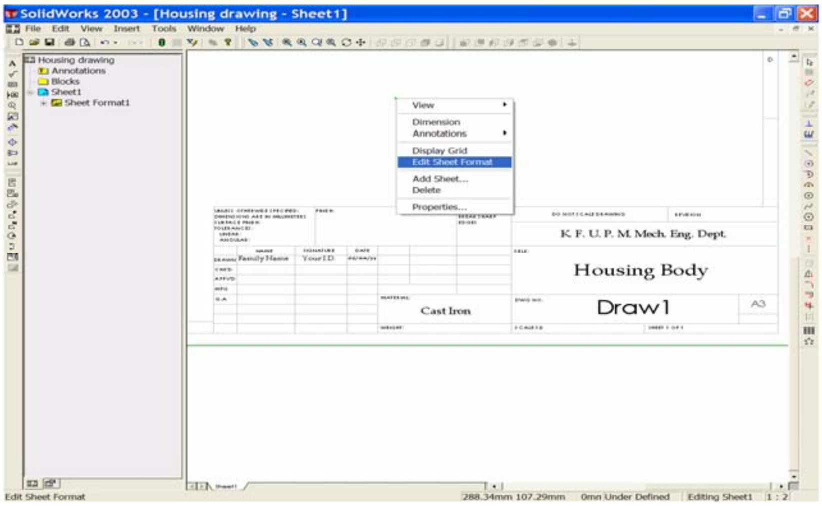


Figure 9

14. Right click in the drawing area and select '**Edit sheet**' once again to exit the command.

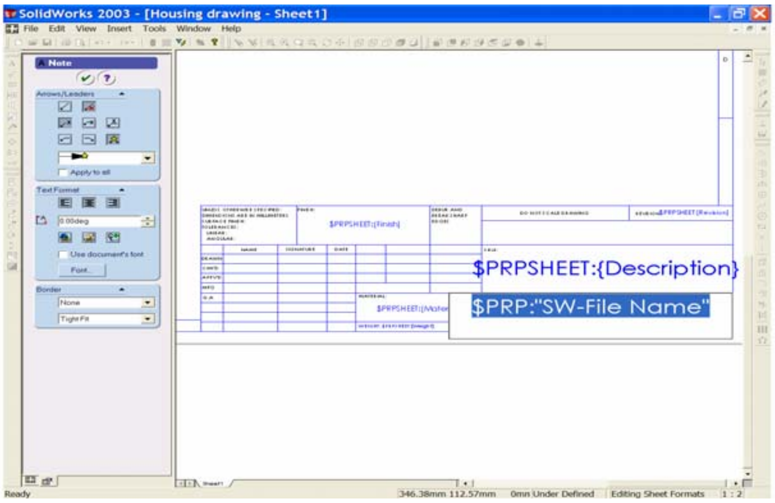


Figure 10

Save your work!

15. The basic preparation is complete so now choose '**Zoom to fit**'. (At this stage the drawing border may be saved as standard A3-Landscape template for further use.)

16. *We are now ready to insert the drawing.*

Select '**Standard 3 View**' from the '**Drawing Toolbar**'. See Figure 11.

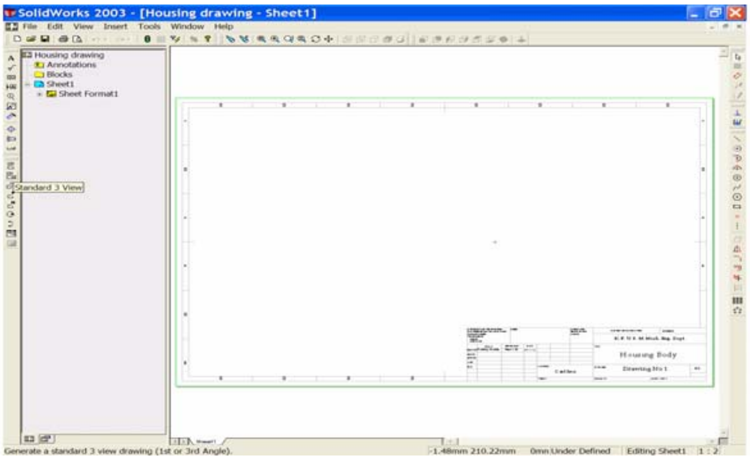


Figure 11

*The '**Standard View Property Manager**' opens and the cursor is accompanied by an isometric view symbol. The Standard View Property Manager lists four different methods you may use to select a model. In this exercise the model file is open, so we can use either choice 2 or 3. See Figure 12.*

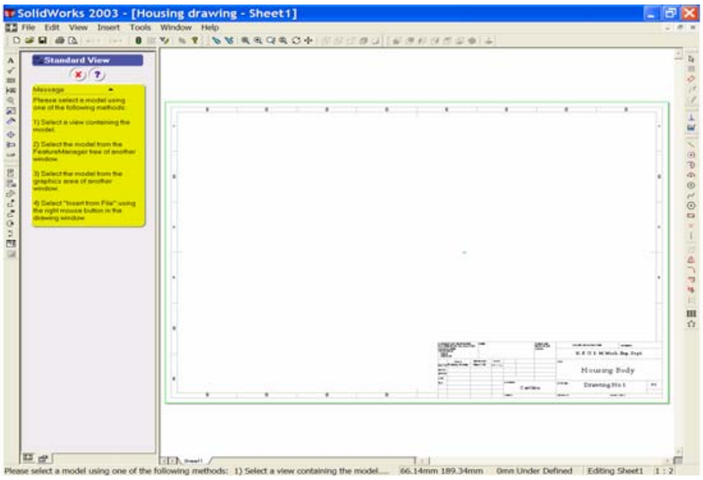


Figure 12

From the window drop down menu, select the Housing Model file. See Figure 13.

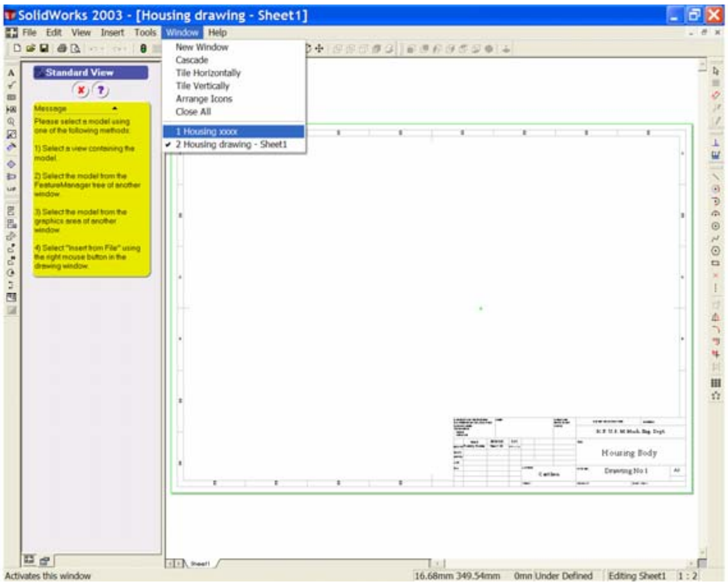


Figure 13

The model window opens with the cursor still indicating the draw symbol.

Left click anywhere in the graphics area of the screen.

The drawing border becomes visible again. We now have to decide if we wish to show 'tangent edges'.

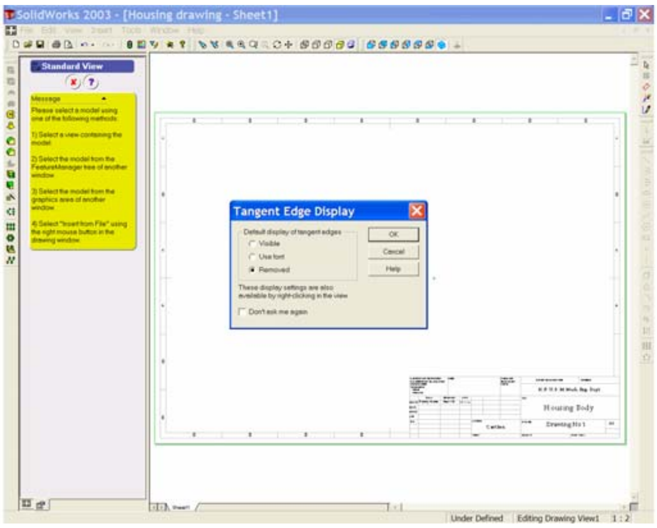


Figure 14

For most circumstances it is best to remove these tangent edges when working with orthographic drawings, so ensure that the '**removed**' option is selected and then select **OK**. See Figure 14.

The Three standard orthographic views of the model are inserted into the drawing sheet border. These views are added to the Feature Manager Design Tree. There will be one view highlighted at all times, and we can work on the highlighted view only. See Figure 15.

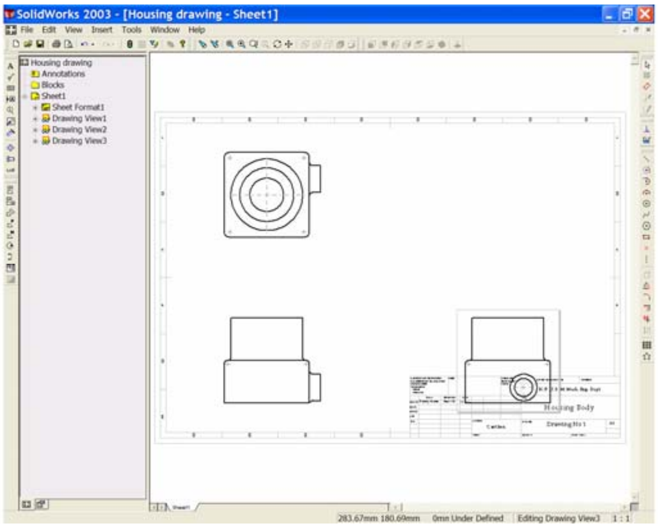


Figure 15

To highlight other views, simply move the cursor close to the view you wish to work with and left click.

17. Place the cursor close to the **PLAN** (the top view).

Notice that the view is framed in a grey box and that the cursor now has a view edit symbol next to it.

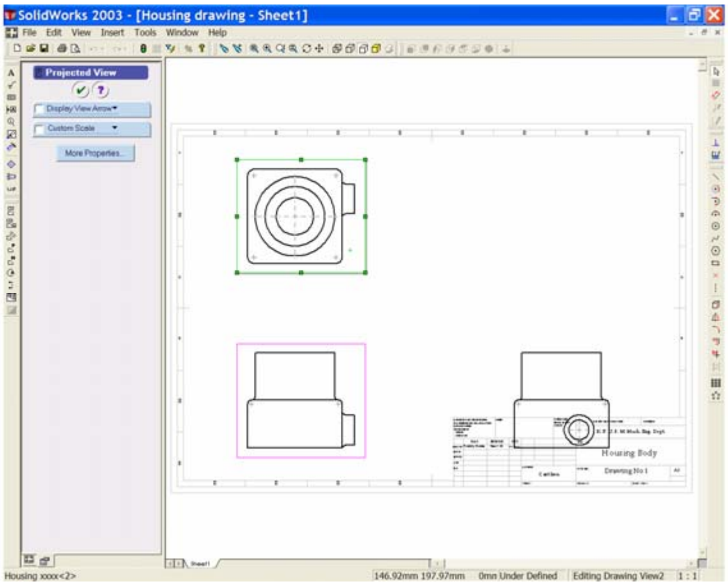


Figure 16

With the cursor inside this grey border left click once. **The Projected View Property Manager** opens and the view is now surrounded by a green window. See Figure 16. Place the cursor on the lower horizontal edge of this window. Note the arrow symbol that appears. **When the arrows are visible press and hold down the left button of the mouse.** We can now move this view 'up' **or** 'down' on the screen. Move the

PLAN View closer to the **ELEVATION** (front view). Note that while we can move this view vertically, we are unable to move it horizontally! See Figure 17.

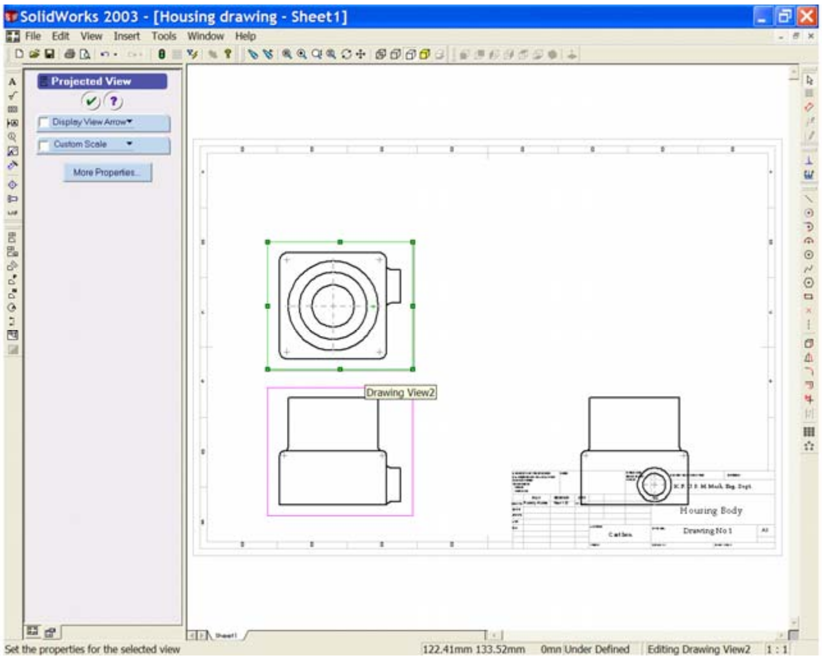


Figure 17

18. Place the cursor close to the **RIGHT END ELEVATION** (right end view) to highlight it. Left click once to activate the window. This time place the cursor on the left vertical edge of the green window. **Press and hold the left mouse button as before and move the view.** Notice that this view can only be moved from '**left to right**'. Place the view at a suitable location to the right of the **ELEVATION**.

19. Now bring the cursor close to the **ELEVATION** (the front view) and activate its window. Place the cursor on any part of the green window, **click and hold the left button once more. See Figure 18. Now, when we move the ELEVATION all three views will move together.** Note also that we can move the orthographic group of views to any location on the screen. Re-position the views so that they do not overlap the drawing border or the title block area. Left click anywhere in the drawing area to end the command.

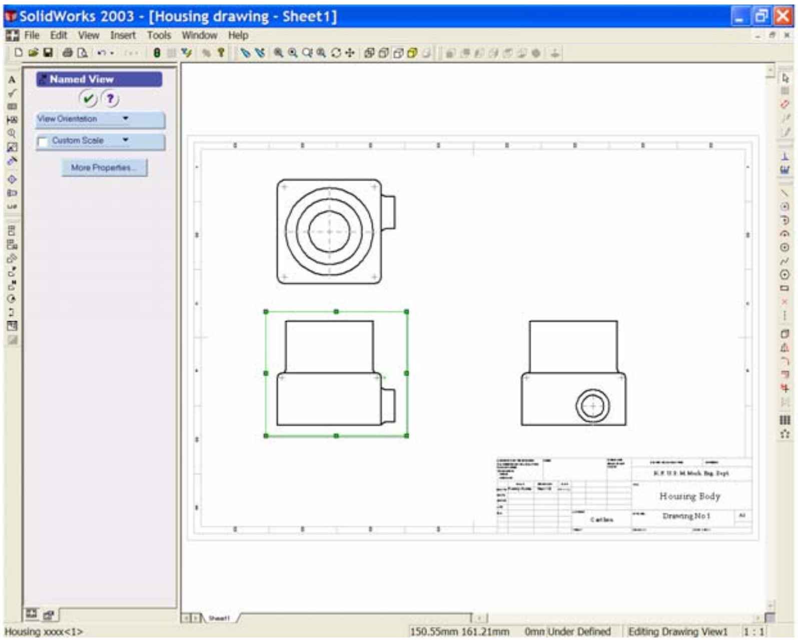


Figure 18

20. We need to show the hidden edges in these views. Place the cursor close to the **PLAN** and left click once. From the view Toolbar choose '**Hidden Lines Visible**'. The hidden edges of the **PLAN VIEW** are now switched on.

21. Place the cursor close to the **ELEVATION** to activate it. Hold down the control button and select the **RIGHT END ELEVATION**. Now switch on the hidden edges for these two views as before.

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Notice that where we can see circles (in the PLAN and the RIGHT END ELEVATION) the centre lies have been added automatically. However we MUST SHOW CENTRE LINES for all cylindrical features of the orthographic views.

22. To add centre lines to the cylindrical features of the drawing, first select '**Centrelines**' from the '**Annotations Toolbar**'. See Figure 19.

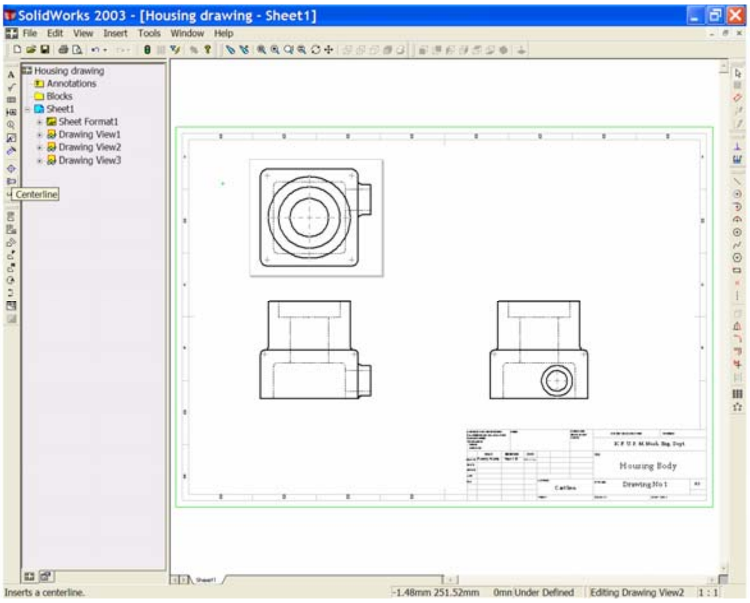

Figure 19

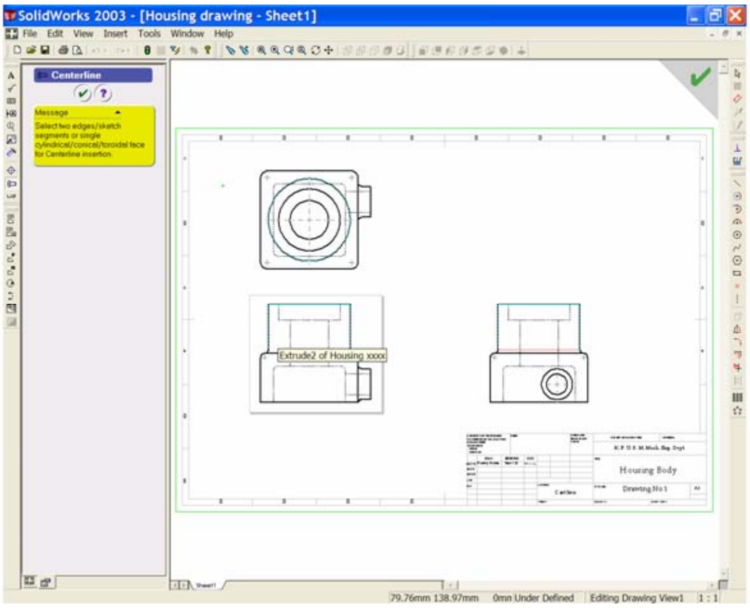

Figure 20

*Notice the message on the **Centreline Property Manager**. This message indicates how we may ad centrelines to the drawing views.*

Place the cursor on the surface of the large boss in the **ELEVATION**. See Figure 20.

Notice the surface flag as the cylindrical surface is identified.

Left click once to add the centreline. This centreline should extend to the bottom of the hidden hole. To lengthen the centreline place the cursor on the centreline, note the colour change and the centreline symbol. Left click to select the centreline. The centreline is now active, is green and has two 'handles' one at each end.

Select the lower handle by pressing and holding the left button of the mouse. Still holding the left button, drag the centreline down to the bottom of the hole and release. See Figure 21.

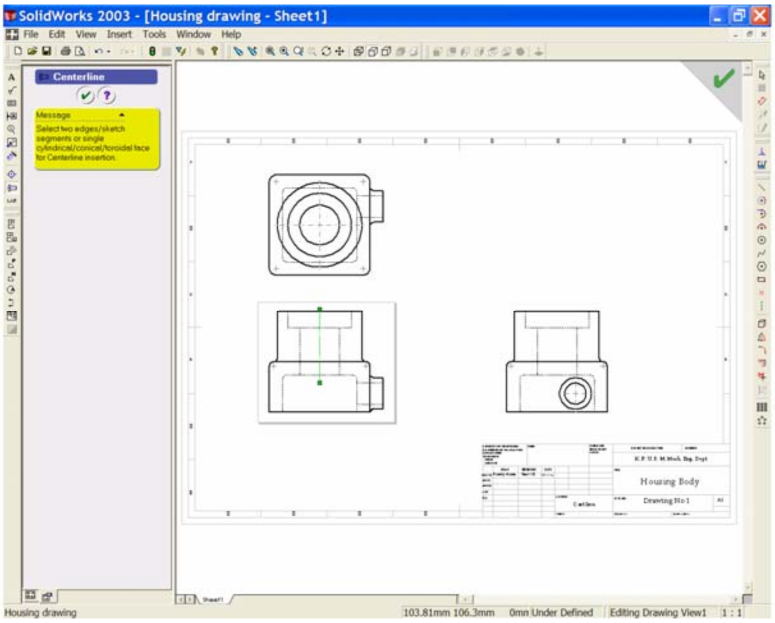

Figure 21

The centrelines of the circles (representing the bosses and bores) in the **PLAN** and **END ELEVATION** may be dragged in the same way.

23. We can place centrelines by identifying the '**edges**' of cylindrical features. Re-start the centreline command. This time select the two edges of the small bore in the **ELEVATION**.

*Note the symbol indicating that you have placed the cursor on the correct '**edge**'.*

Select the edge, it will turn green; (See Figure 22) continue to select the opposite edge of the bore. Once again the symbol indicates that the cursor is on the bore '**edge**'. Select this second edge to complete the command. The centreline is placed.

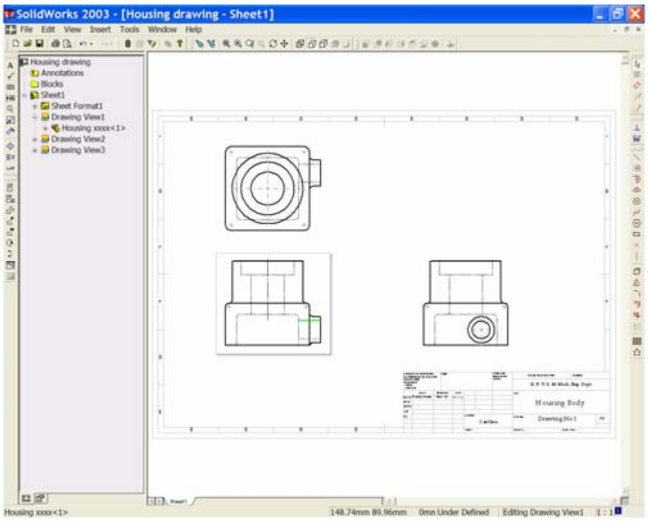

Figure 22

Repeat the above steps for the small bore in the **PLAN**. When you are finished, choose the green tick to close the Centreline Property Manager.

24. Some section views would be useful for the drawing of this part. We will use the '**Broken-out**' section in two places.

From the '**Sketch Toolbar**' choose '**Circle**'. The pencil/circle symbol appears on screen and we will now draw a circle to establish the area for the broken-out section. Place the cursor pencil/circle symbol as indicated in Figure 23 and draw a circle as shown below.

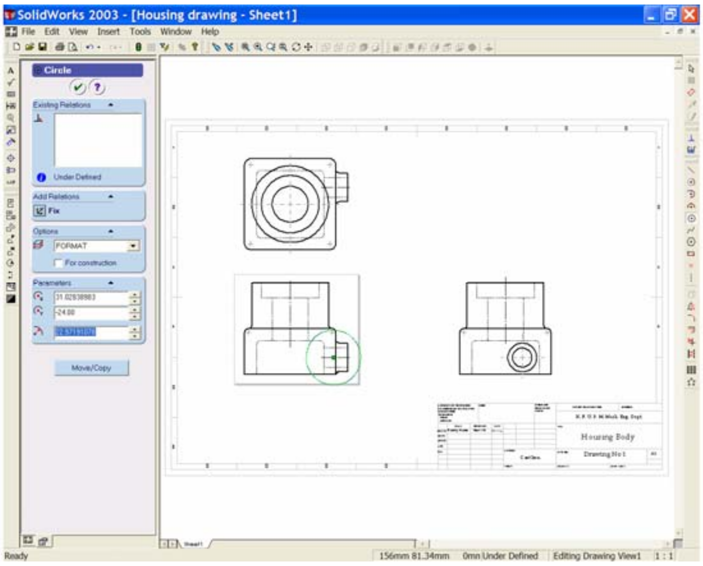

Figure 23

25. From the '**Drawing Toolbar**' select '**Broken-out Section**'. See Figure 24.

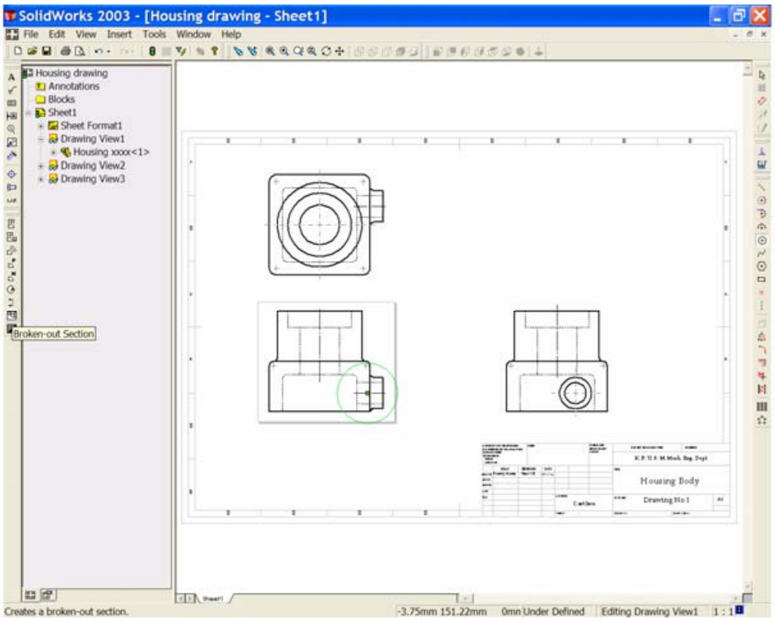

Figure 24

The Broken-out Section Property Manager opens.

Check the '**Preview**' box. If a message box opens on your screen choose **OK**.

Notice that a yellow cutting plane is shown on the PLAN and RIGHT END ELEVATION. See Figure 25.

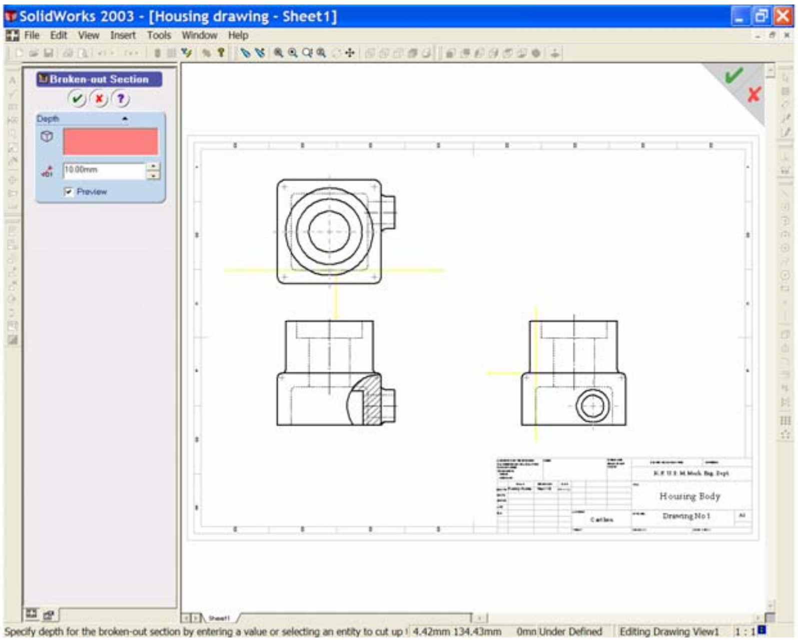

Figure 25

The Property Manager indicates a default depth of 10mm. Press the up arrow to increase the depth to 20mm.

Notice that the yellow cutting plan moves by 10mm and that the broken-out section now appears inside the circular area.

Press the up arrow and increase the depth to 50mm, watch the cutting plane move and the effect it has on the broken-out section. The cutting plane is now close to the centre of the small boss & bore. See Figure 26.

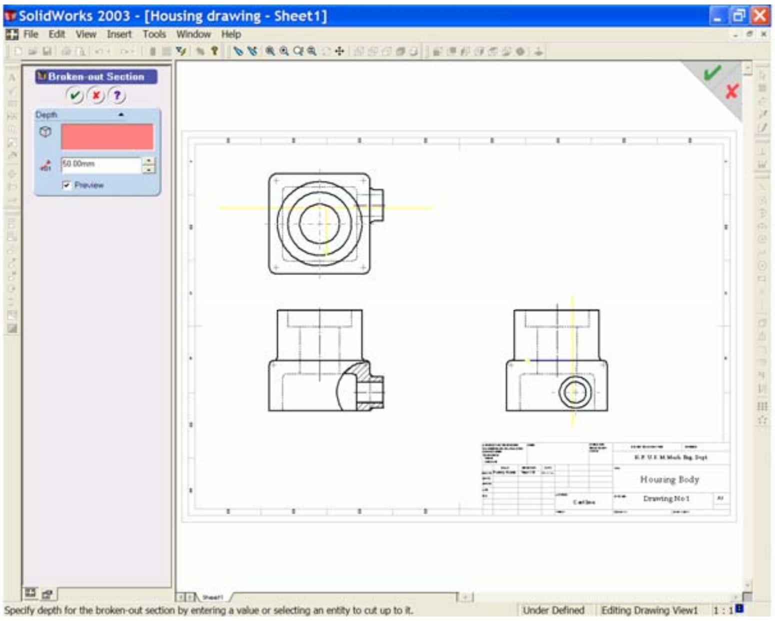

Figure 26

We want the cutting plane to pass through the boss & bore, exactly on its centre. To achieve this we may select the circle that represents the diameter of the boss. Place the cursor on the boss circle.

Notice that it is highlighted; there is an edge symbol next to the cursor and a note indicating the feature (Extrude 3 of Housing). See figure 27.

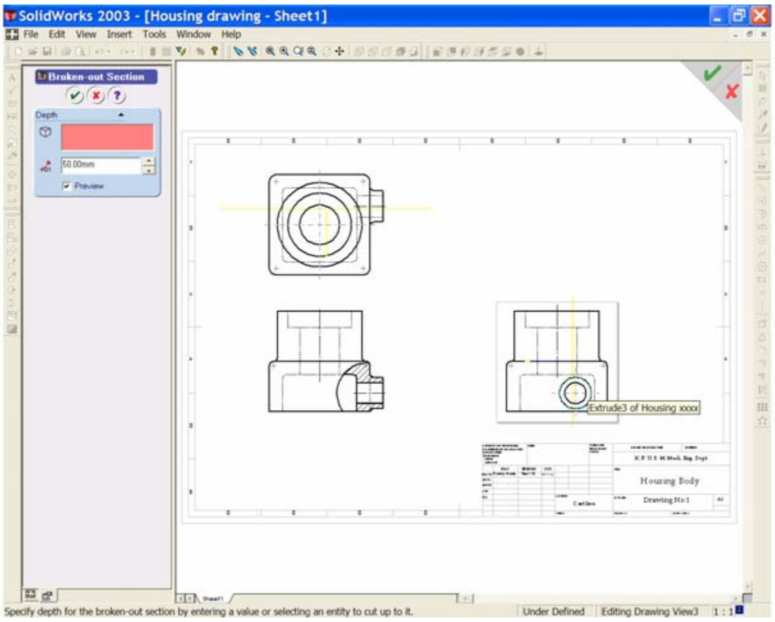

Figure 27

Left click to accept the boss feature. The yellow cutting plane is now exactly on the boss/bore centre, the broken-out section is complete and Edge<1> is listed in the depth window of the Broken-out Section Property Manager. See Figure 28.

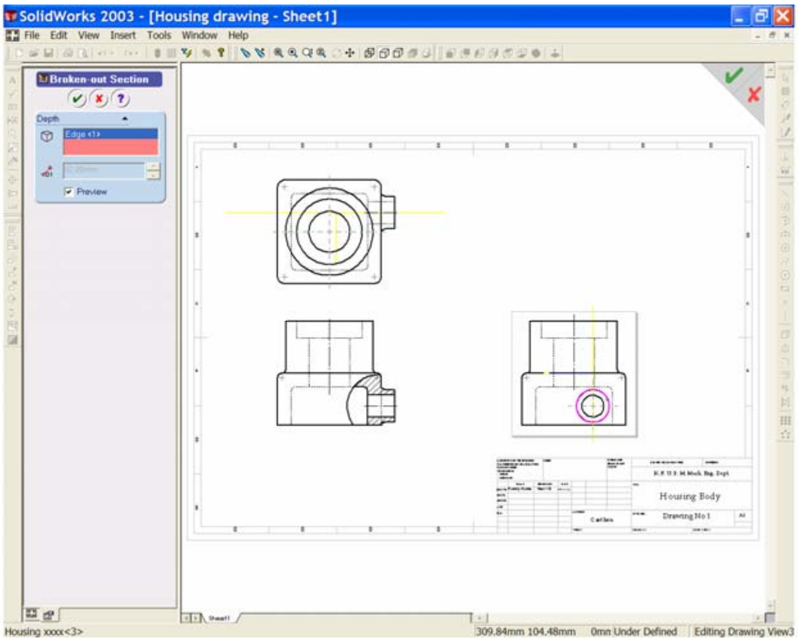

Figure 28

26. Select the green tick  to end the command.

27. On your own, use the same procedure to create a second Broken-out Section for the counterbore hole. See Figure 29.

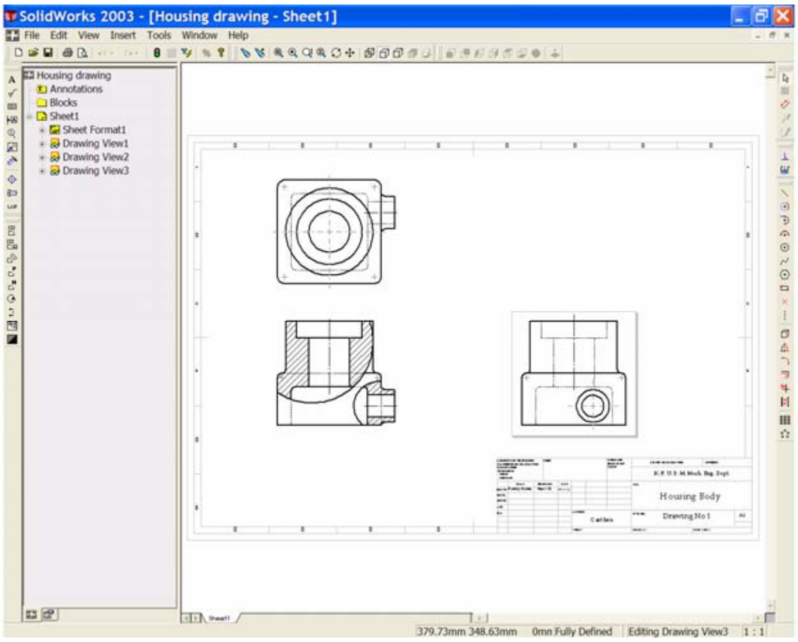

Figure 29

Notice that the broken-out sections are added to the design tree.

28. In the section view, normally hidden edges are not shown. Select the **ELEVATION** and switch off the hidden edges. See Figure 30.

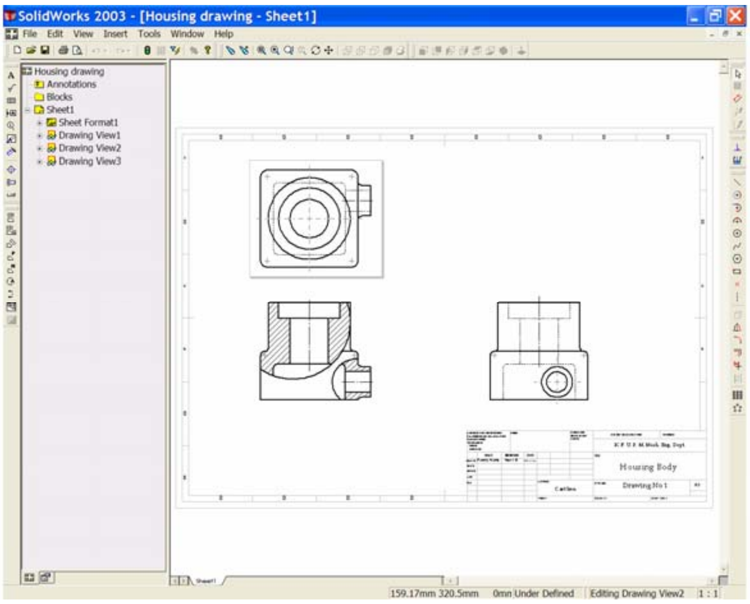

Figure 30

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29. We are now ready to insert the dimensions. Place the cursor over the '**Insert**' dropdown menu at the top of your screen and open. Choose '**Model Items…**' See Figure 31. '**The Insert Model Items**' dialog box opens. Ensure that the '**Dimensions**', '**Import items into all views**'

and '**Eliminate duplicate model dimensions**' boxes are checked. Choose **OK** to insert the dimensions. See Figure 32.

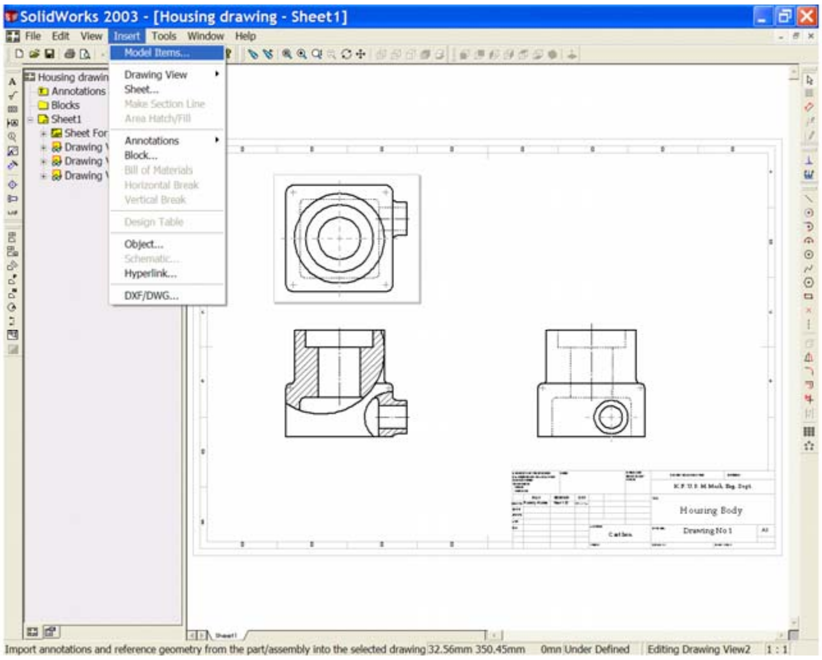

Figure 31

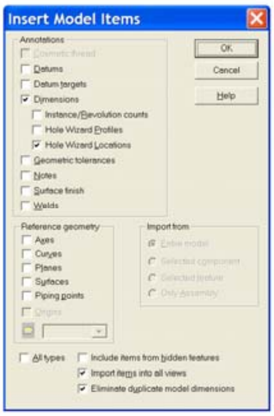

Figure 32

Note that the dimensions are not arranged well. The software simply dumps the dimensions from the model into the drawing file. See Figure 33.   ***We must now re-arrange the dimensions to conform to International Standards.***

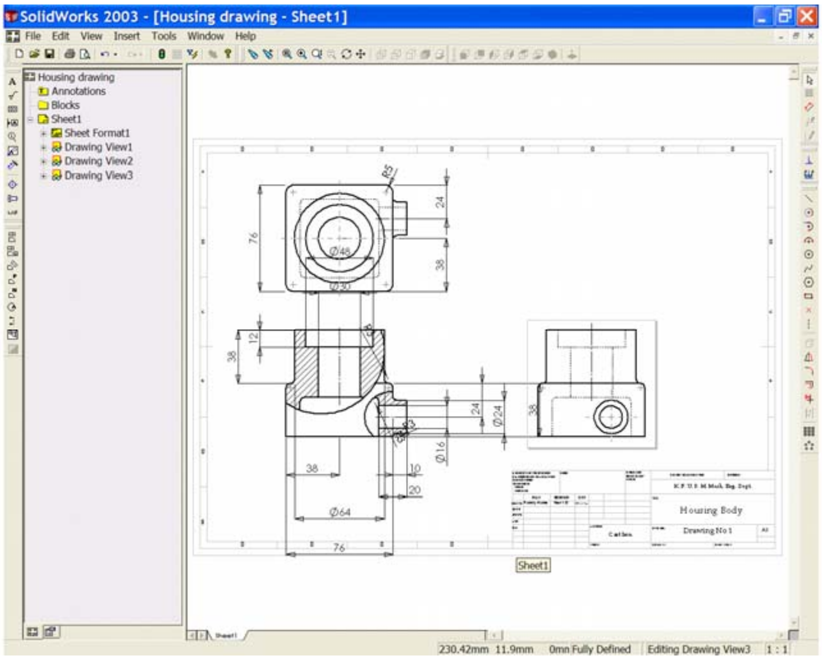

Figure 33

30. First, we should hide dimensions that may be given in a note. The most suitable note dimension is the R3 fillet. **Place the cursor over one of the R3 fillet dimensions and right click.** See Figure 34.

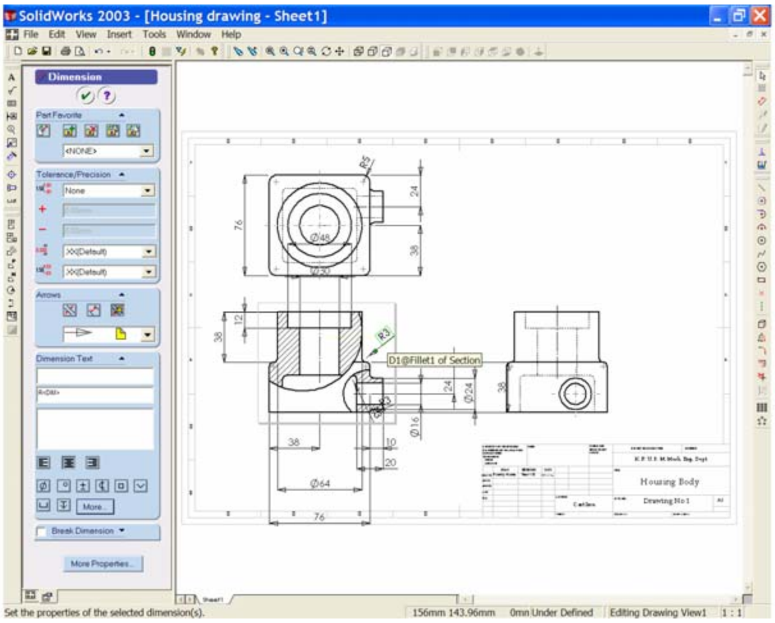

Figure 34

From the short-cut menu choose '**Hide**', the dimension is hidden from view. Continue to hide all other R3 fillet dimensions. See Figure 35.

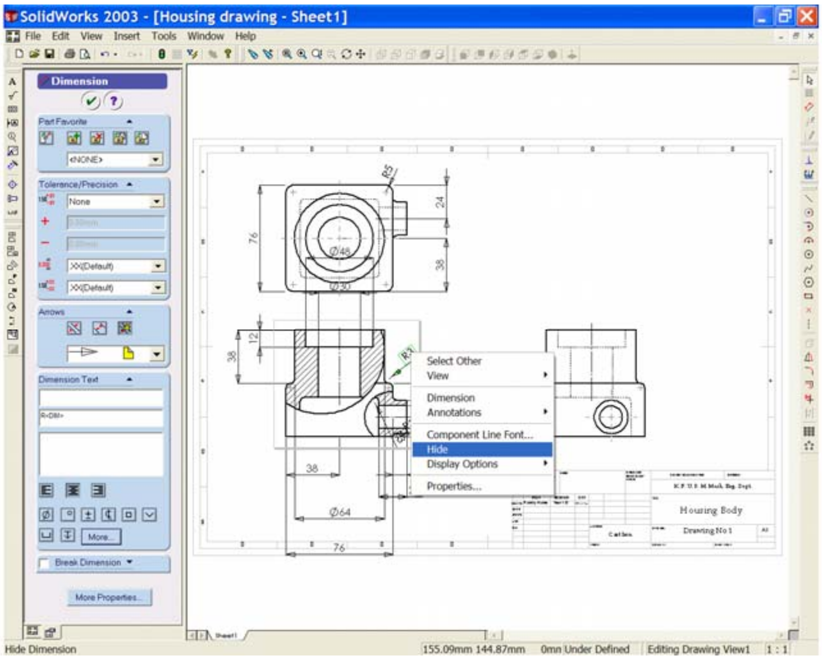

Figure 35

31. The location dimensions of the small boss/bore are best shown in the **RIGHT END ELEVATION**, therefore the dimensions should be moved from their present locations to this view. To do this hold down the control key as you select the dimension with the left button of the mouse, then drag the dimension to the **END ELEVATION**. Drop the dimension at the location shown in Figure 36.

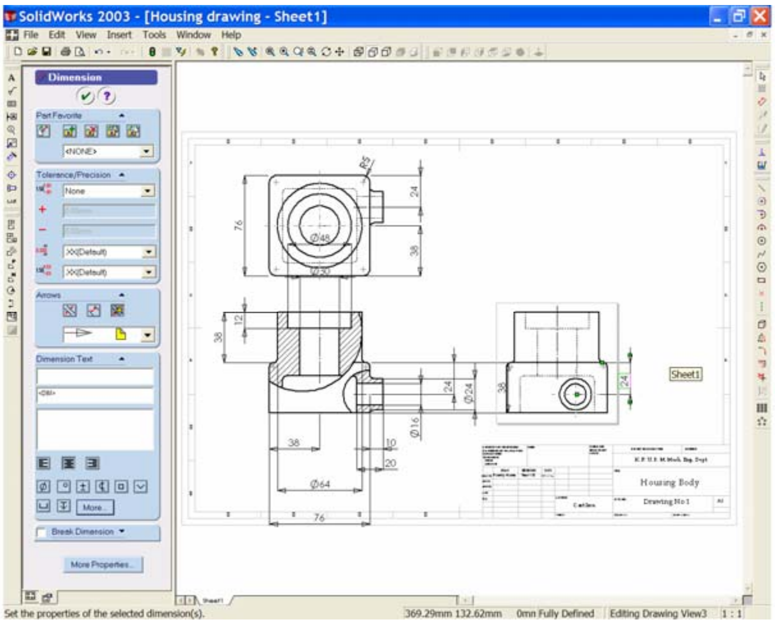

Figure 36

Notice the symbol change as the cursor is moved from the original view to the new location.

32. Other dimensions may be moved to better locations within a view by selecting them with the mouse, left clicking and hold as the dimension is dragged to a new position. When finished, your drawing should look similar to Figure 37.

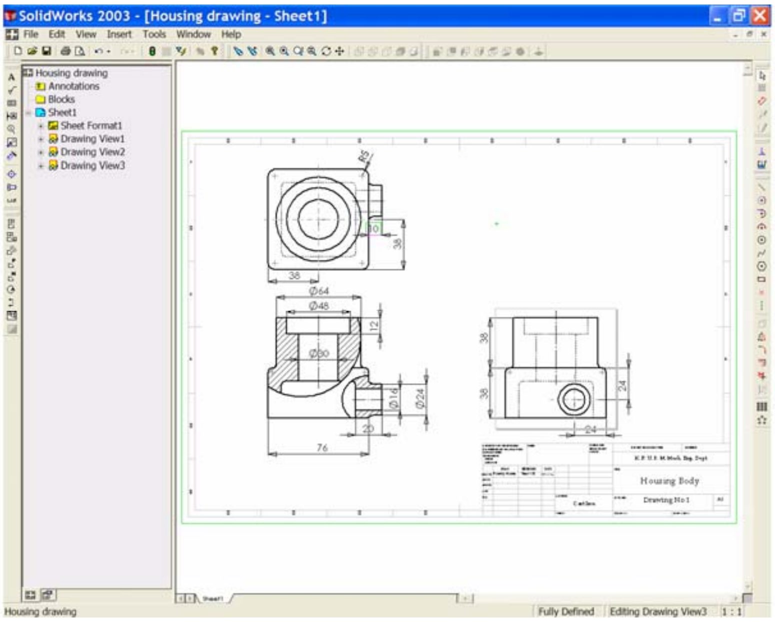

Figure 37

33. When modifying dimensions in the drawing it will be necessary to save both the drawing and its referenced model. When this must be done a dialog window will open asking if you wish to save the referenced models. When this happens it is best to answer '**yes**'. Both the active drawing and its referenced model will be saved.

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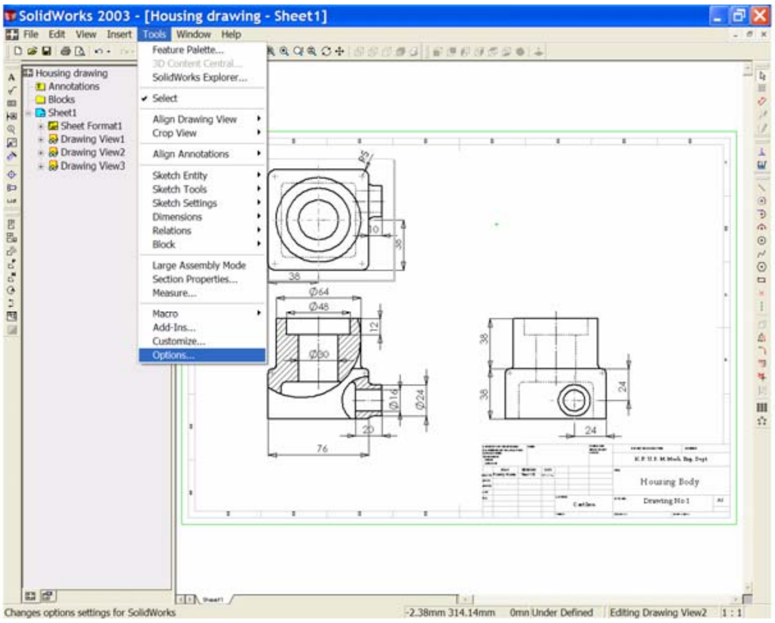


Figure 38

34. The dimension font, font size, arrow style and arrow size may be changed globally. From the '**Tools**' drop down menu choose '**Options…**', See Figure 38. The System Options dialog box opens, usually at '**General**'. Select the '**Document Properties**' tab. Under '**Detailing**' select '**Dimensions**'. See Figure 39. In the '**Arrows**' area, choose the down arrow to view and select an arrow style.

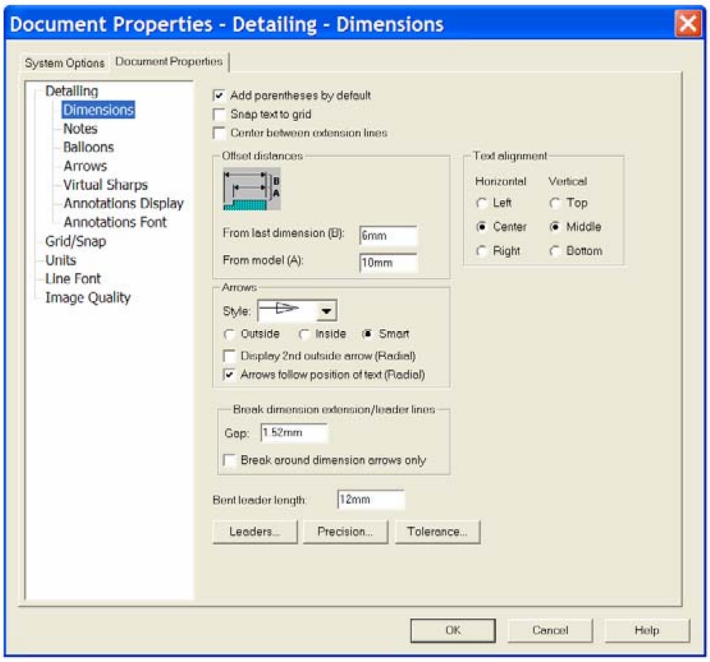


Figure 39

35. Choose '**Annotations Font**' under '**Detailing**' and from the annotations type window select '**Dimension**'. From the dialog box select '**Book Antigua**' and set the **Height to 12 points**. Select **OK** to close the box. See Figure 40.

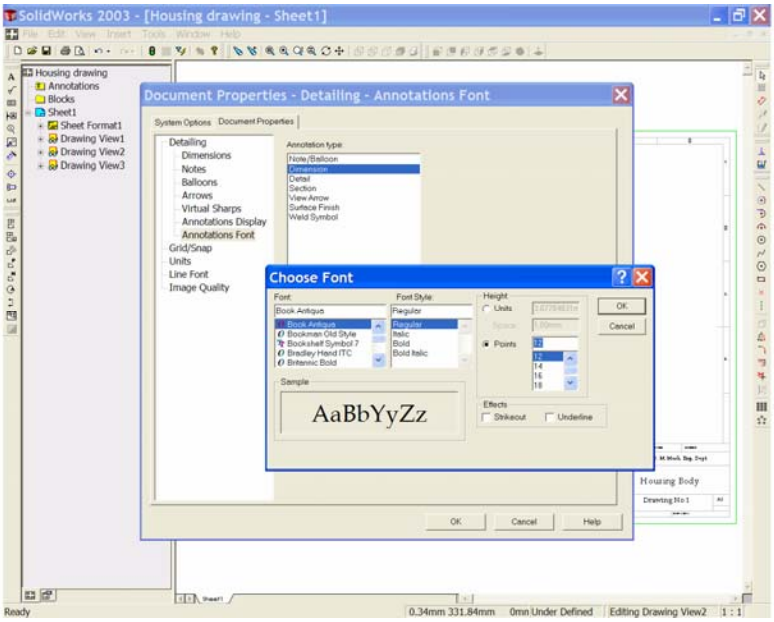


Figure 40

The drawing dimensions and arrows should now have the size and font selected. See Figure 41.

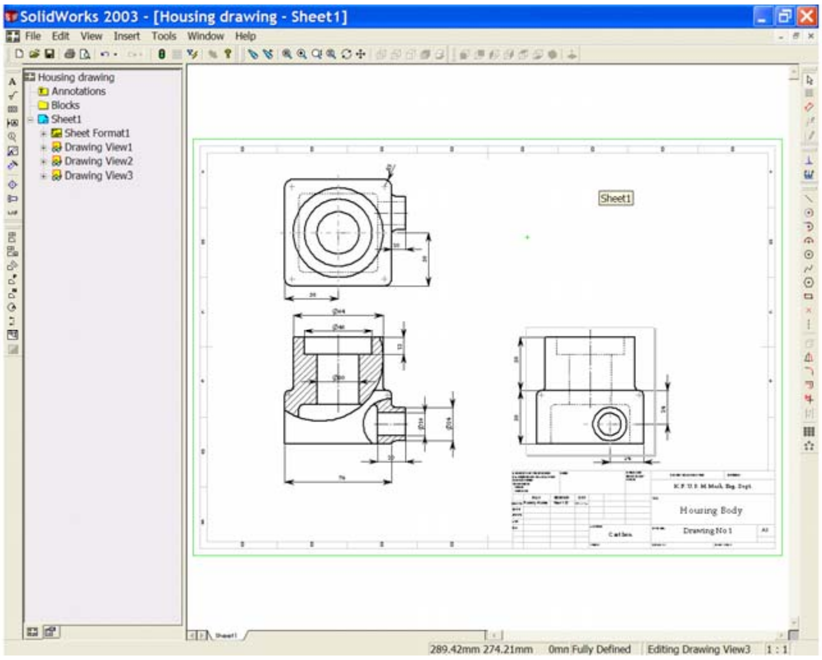


Figure 41

36. We should now add the note to give the dimension for the rounds & fillets that we have hidden. Choose '**Note**' from the '**Annotations Toolbar**' as before. Place the note box in a suitable location (See Figures 42 and 43.) and type '**Rounds & fillets to be R3 Unless otherwise stated**'

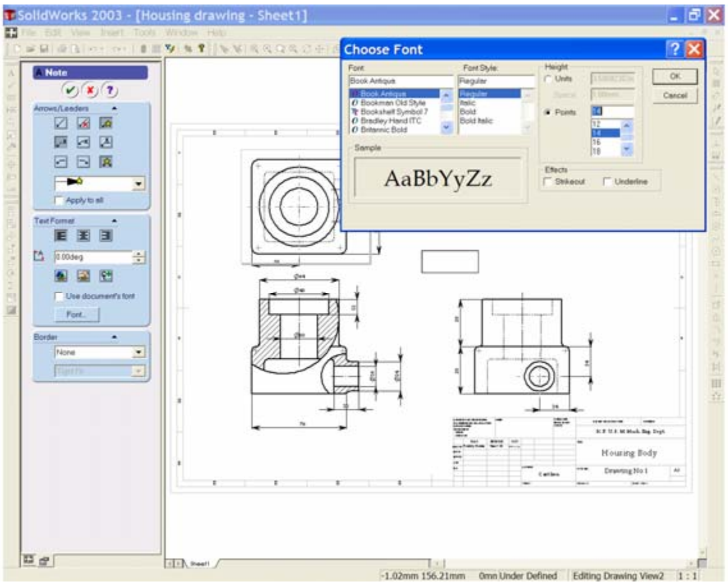


Figure 42

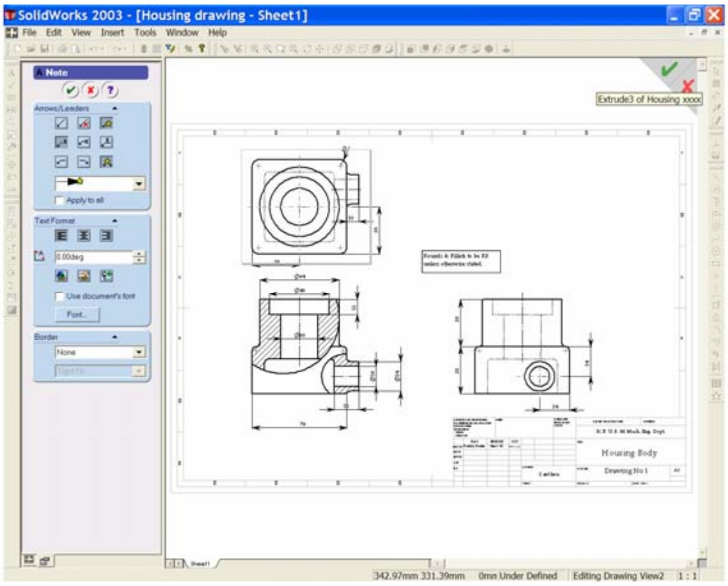


Figure 43

37. To finish we may now add a pictorial view of the model. From the '**Drawing Toolbar**' select the '**Named View**' button. See Figure 44. The Named View Property Manager opens and indicates four methods of selecting the model. See Figure 45. We will use method 1 for this drawing.

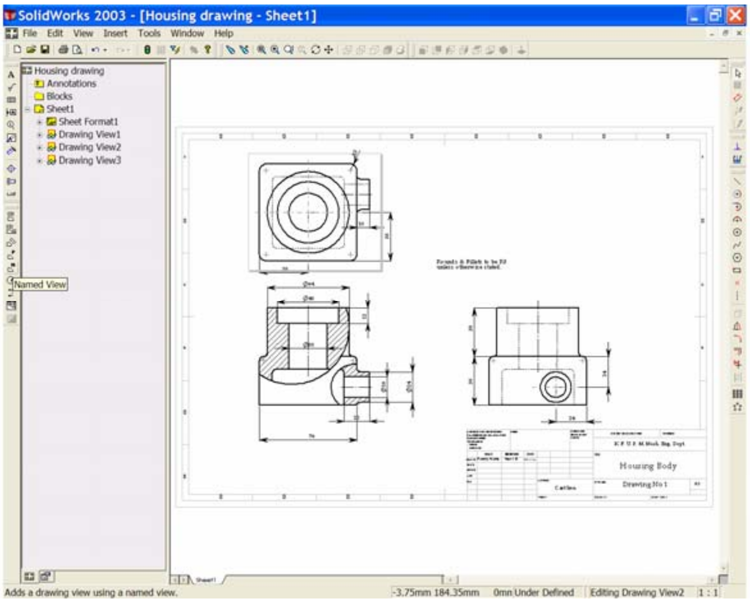


Figure 44

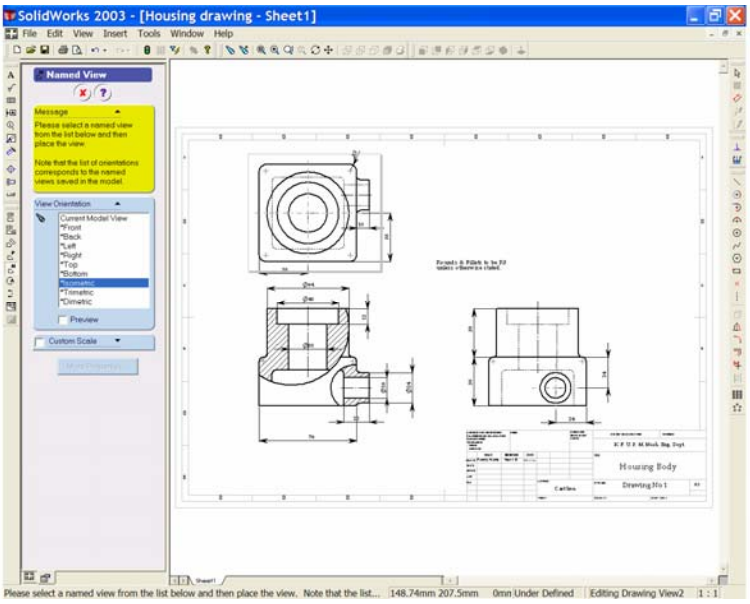


Figure 45

38. Place the cursor close to the **PLAN** view and left click once. A view orientation window opens in the Property Manager and the cursor changes to indicate that a named view may now be selected from the list offered. In the window ensure that '**ISOMETRIC**' is highlighted, you may also check the preview box. Place the isometric view in a suitable location at the top right corner of the border. (You may experiment with some other choices such as Trimetric or Dimetric.)

    When you have placed the view choose the green tick to complete the command. See Figure 45.

    *Notice that the '**tangent edges**' are visible in the isometric view. These edges may be displayed with different types of lines.*

39. Right click close to the **ISOMETRIC VIEW** and from the short-cut menu select '**Tangent Edge**'. A further menu slides out to the side and offers three choices; '**Tangent Edges Visible**', '**With font**' or '**Removed**'. Choose '**Tangent Edges With Font**'. Your model will now display the Isometric view tangent edges as phantom lines. See Figure 46.

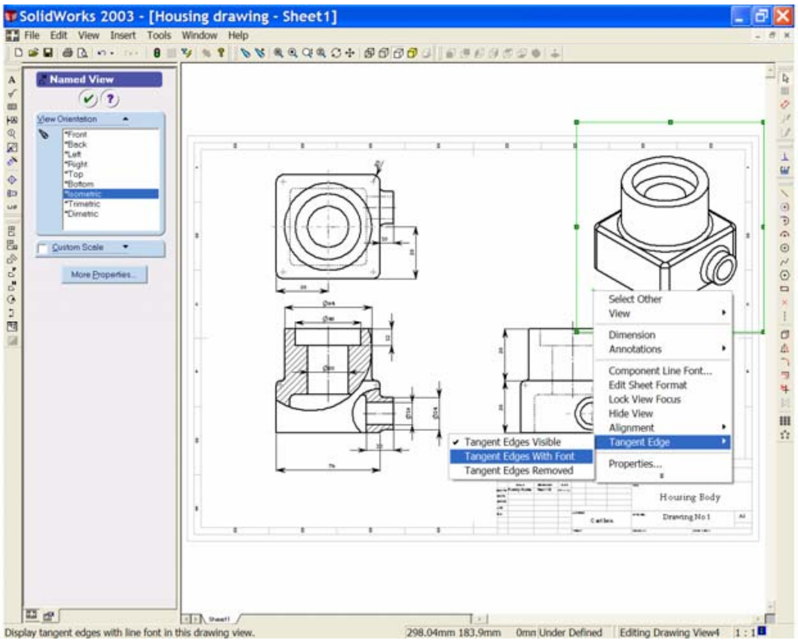


Figure 46

40. Finally, we may add colour to the Isometric view. Left click close to the isometric view and select '**Shaded**'  from the '**View Toolbar**'. You may need to left click anywhere in the graphics area to re-generate the drawing.

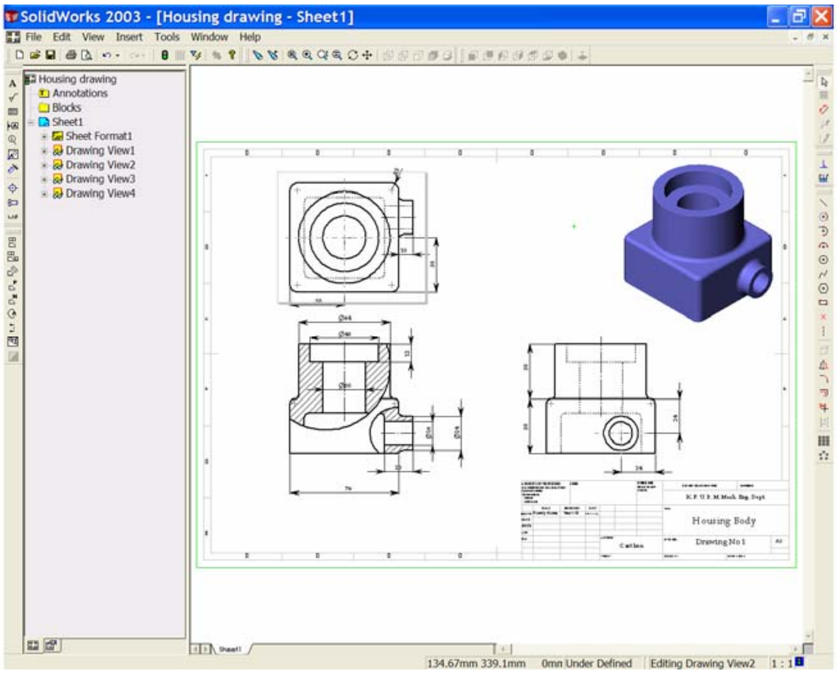


Figure 47 The complete Drawing

Module 6: To Create a 2-D Working Drawing

1. Open the Housing drawing.

2. *The first enhancement we will add to the drawing will be layers*. Figure 1 indicates the route to the layer toolbar. Go ahead and open the layer toolbar. The toolbar may '**float**' or may be '**anchored,** (fixed) at a convenient location on your screen.

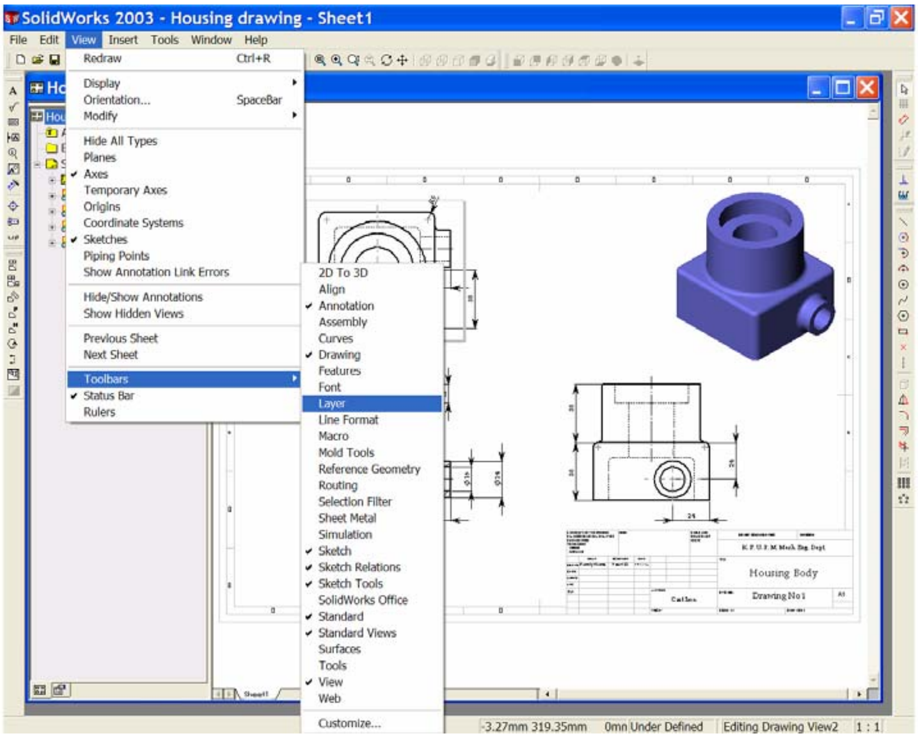


Figure 1

3. Open the layers **Property Manager**, Figures 2 & 3.

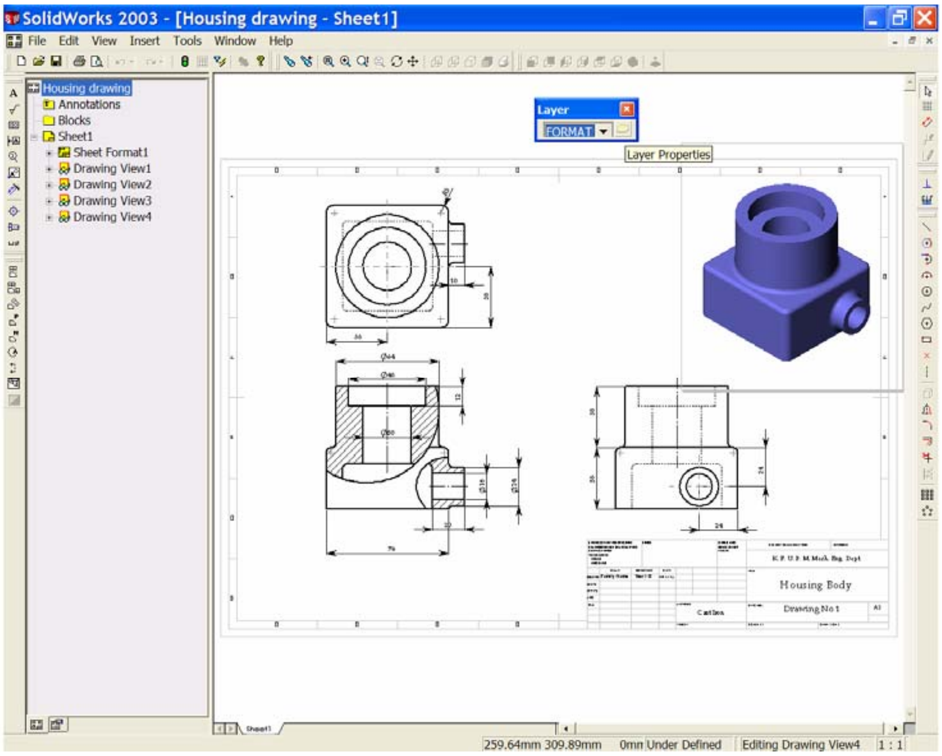


Figure 2

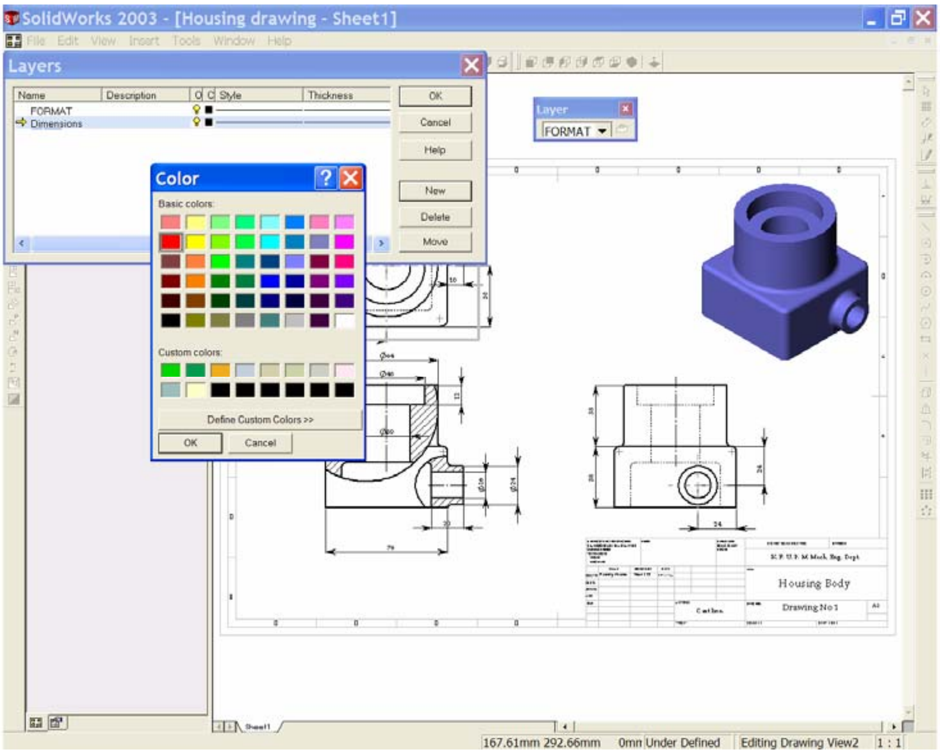


Figure 3

4. In the **Layers Property Manager** create a new layer by selecting the '**New**' button. Give this layer a new name '**dimensions**', and a new colour (you may choose your own colour for the layers).

5. Create two more layers. One called '**Machine Symbols**' and the other called '**Tolerances**'. Give each layer a different colour. See Figure 3.

6. We may now change from one layer to another by using the ‘**down arrow**’ of the layer toolbar.

7. To place the dimensions on the dimension layer. **Holding the control key down**, select each dimension in turn. As each dimension is selected, it is identified on screen by green markers. When all dimensions are selected, use the down arrow of the layer toolbar to select the ‘**dimension**’ layer. The active dimensions are now transferred to this layer and take on the colour you have set. See Figure 4. Select **OK** from the dimensions Property Manager to end the command.

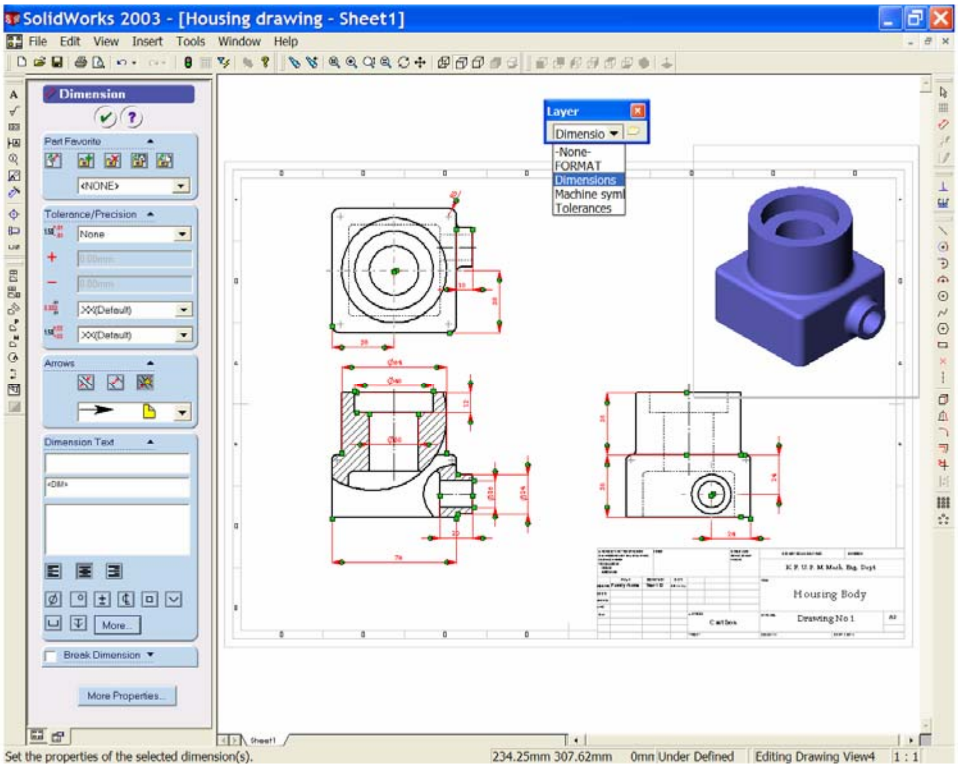


Figure 4

8. Line thicknesses may be changed from the ‘**Tools’**, ‘**Options…’** drop down menu. Open the Systems Options window and select ‘**Document Properties**’. Select ‘**Line Font**’. See Figure 5. From this window we may choose many ‘types’ or ‘styles’ of lines and choose their ‘thicknesses’. Make the following settings in this window;

   i. Visible Edges……… Style: ‘**Solid**’ Thickness: ‘**Thick**’
   ii. Hidden Edges……..... Style: ‘**Dashed**’ Thickness: ‘**Normal**’
   iii. Dimensions……….... Style: ‘**Solid**’ Thickness: ‘**Thin**’

Choose **OK** to close the window and view the results on screen. See Figure 6.

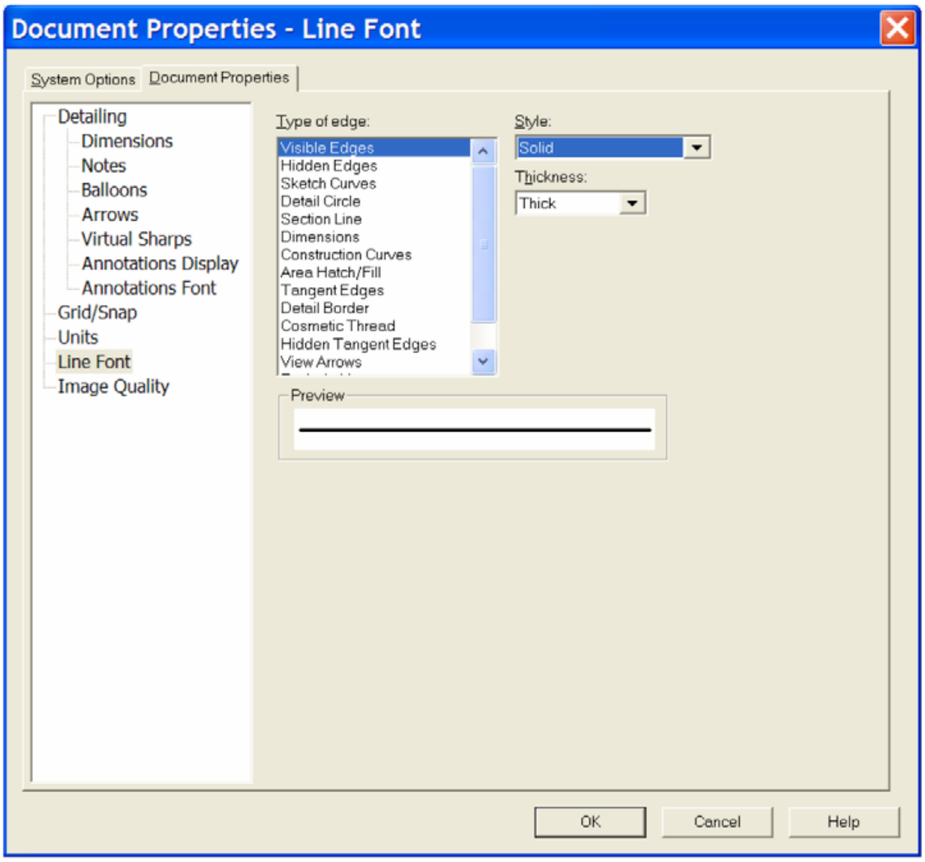


Figure 5

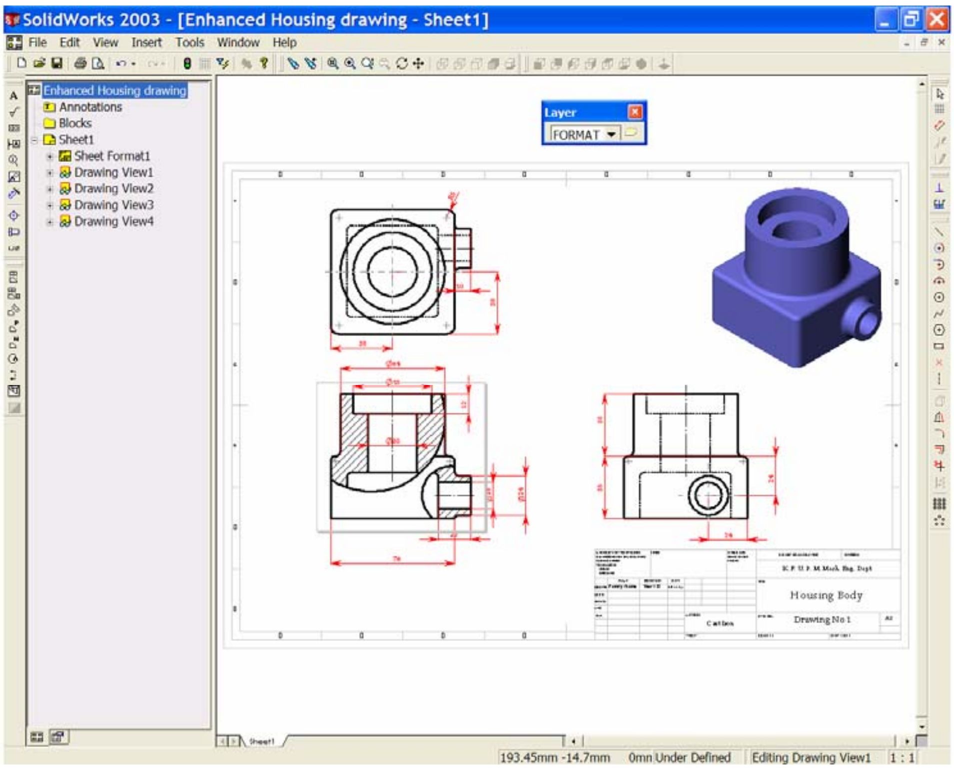


Figure 6

9. **Save your drawing** as ‘Enhanced Housing Drawing xxx’.

10. Three of the diameters require limits. These are the Ø30 bore, the Ø48 bore and the Ø16 bore. The assignment sheet asks for an '**H7 tolerance**' on each of these diameters. To add tolerances to dimensions first, select Ø30 to activate it, See Figure 7, this will open the dimension properties manager. In the '**Tolerance Type**' window choose '**Fit with tolerance**'. From the '**Classification**' window choose '**Press**'. From the '**Hole Fit**' window choose '**H7**'. From the '**Primary Unit Precision**' and '**Tolerance Precision**' windows choose **(.XXX [three decimal places])**. See Figure 8, and select **OK** to complete the command.

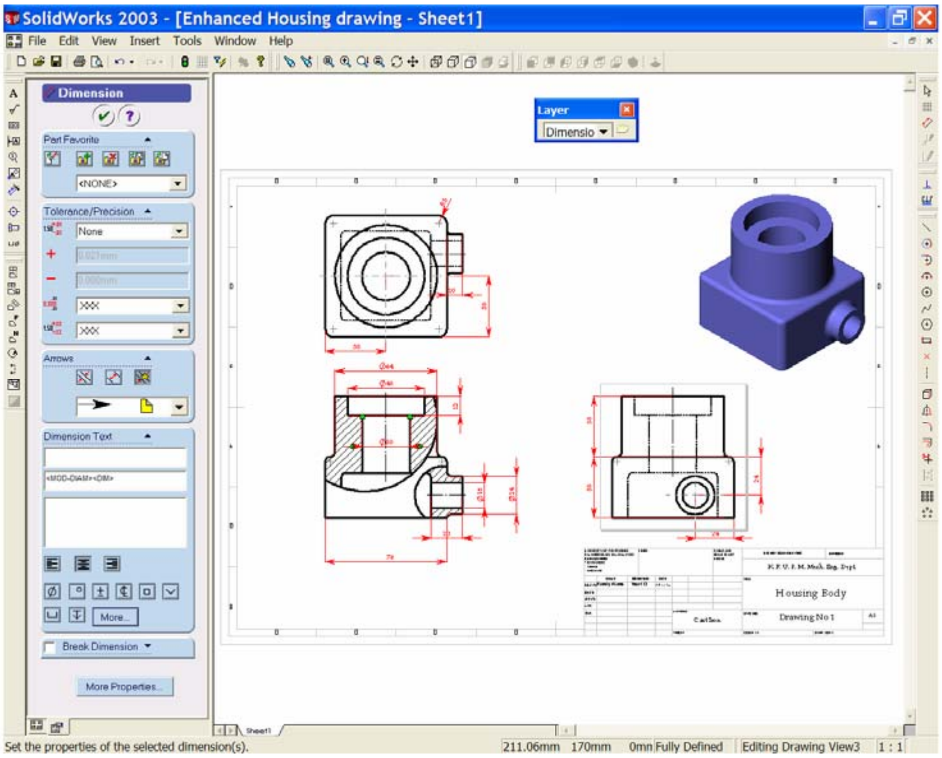


Figure 7

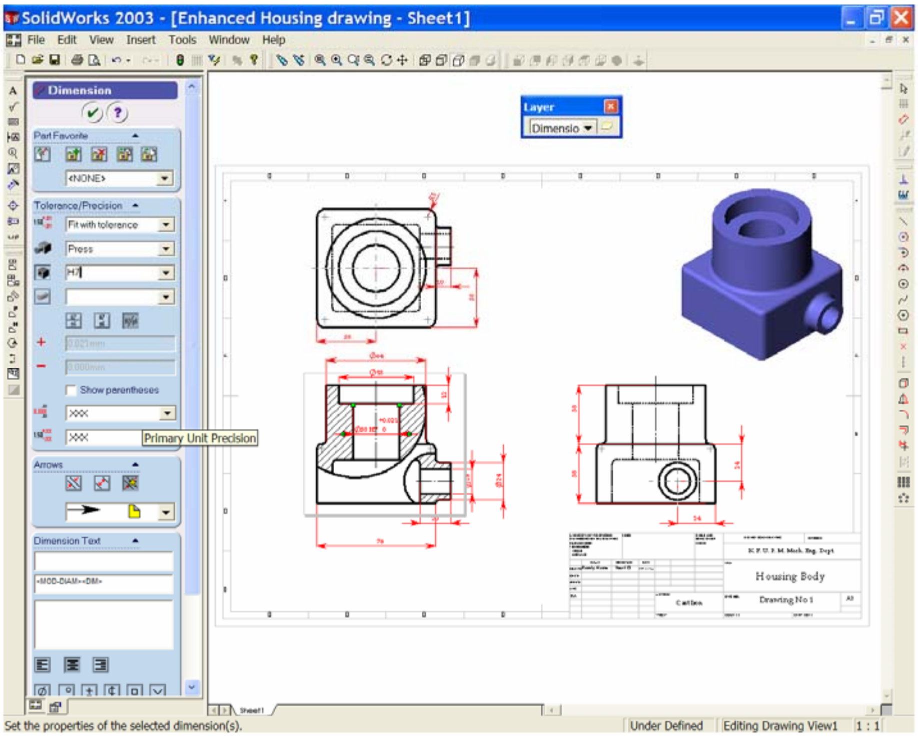


Figure 8

11. Select the Ø30 dimension and place it on the tolerances layer. Notice the change of colour as it moves from one layer to the other. See figure 9.

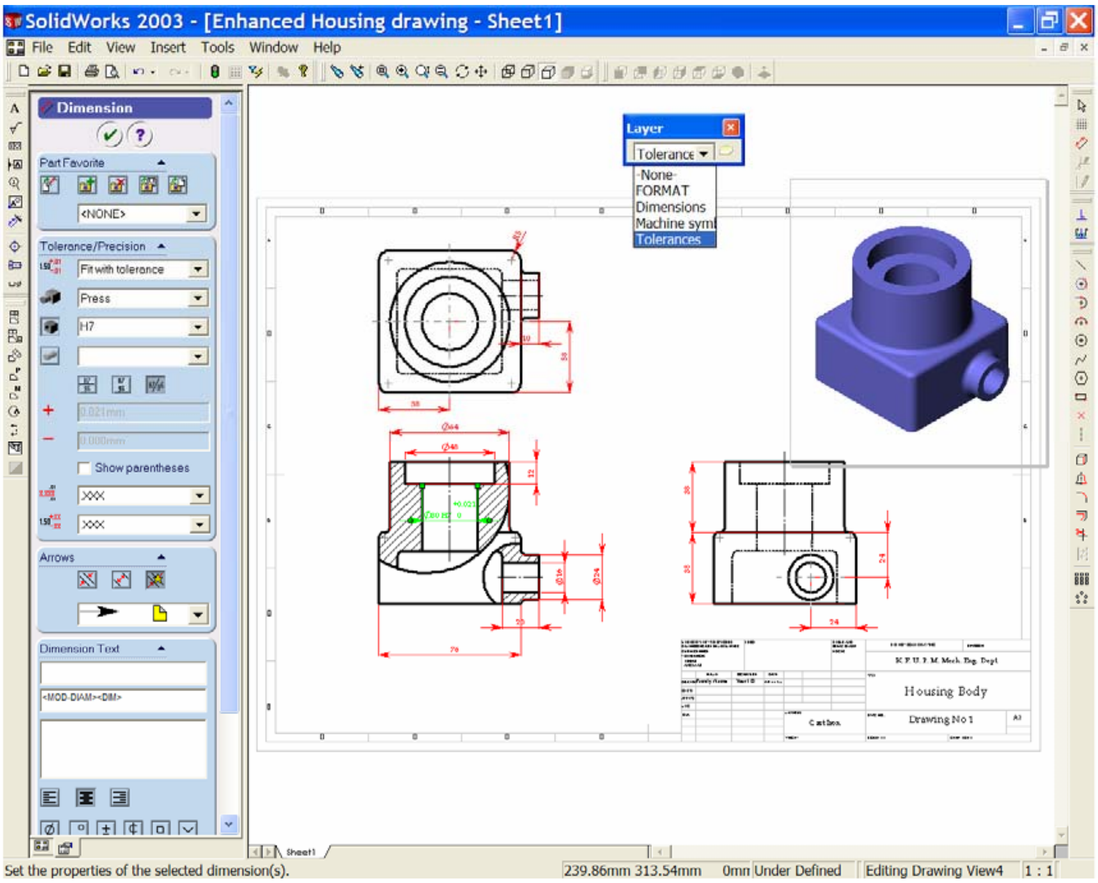


Figure 9

12. Now add an **H7 tolerance** to the other two diameters. When complete, move these two diameters to the tolerance layer.

13. Use the '**Zoom to Area**' tool to look more closely at Ø16. You will notice that the data does not fit well into the available space. We can display the data in several ways. In the '**Tolerance Type**' window select '**Limit**'. Notice that the maximum and minimum limits are now displayed for this dimension, and that it fits better into the space.   Use '**Zoom to Fit**' to return to the full drawing. See Figure 10.

SAVE YOUR WORK!!

14. We need to add machine symbols (surface texture/finish) to the drawing where they are needed. Set the '**Machine Symbols layer'** active. From the '**Annotations Toolbar**' select '**Surface Finish**' See Figure 10.

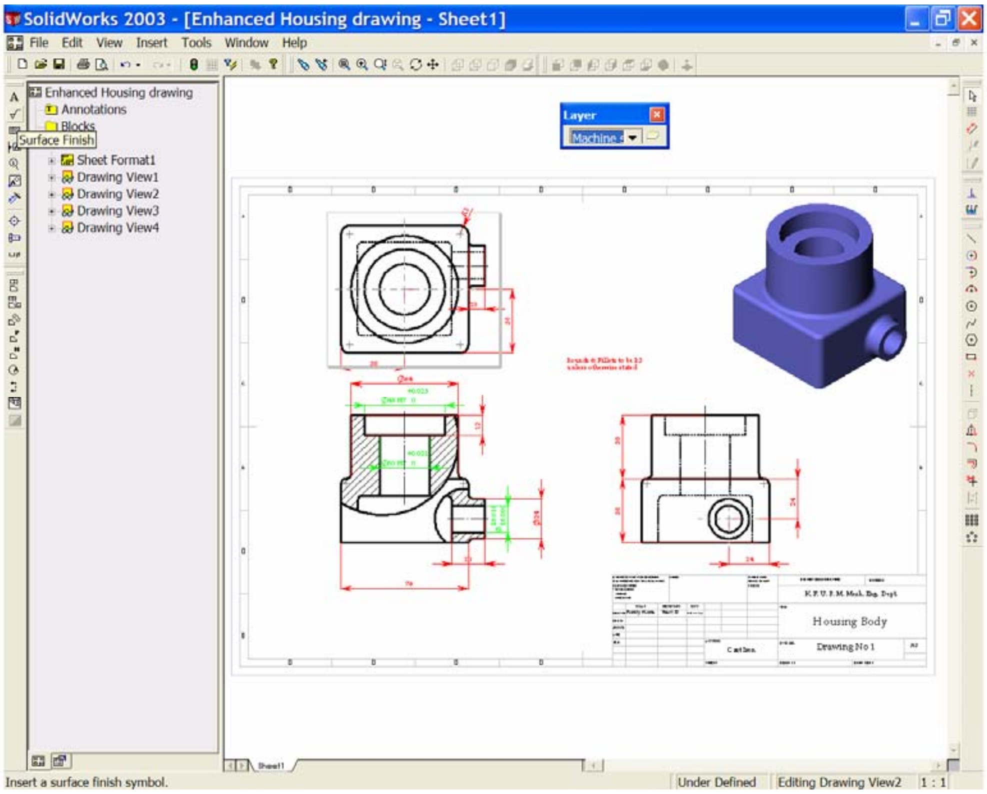


Figure 10

15. The **Surface Finish Symbol Properties** window opens. Notice the preview window. In the '**Symbol Window**' choose '**Machining required**'. Notice the change in the preview window. In the '**Direction of Lay**' window choose '**Circular**'. In the '**Roughness Maximum**' window enter a value of **0.8 (µm)**. In the '**Material removal Allowance**' window enter a value of **3.5 (mm)**. Notice the symbol being built in the preview widow. We **DO NOT WISH** to use **Leader Lines** to place the symbols, so ensure that '**No Leaders**' is selected. We may control the size of the symbol by using the '**font**'. Un-check the '**Use documents font**' box and open the font dialog box. Change the font to '**Book Antigua**' with a height of **12 points**. Notice that the symbol in the preview box shows these changes as they are entered. See Figure 11.

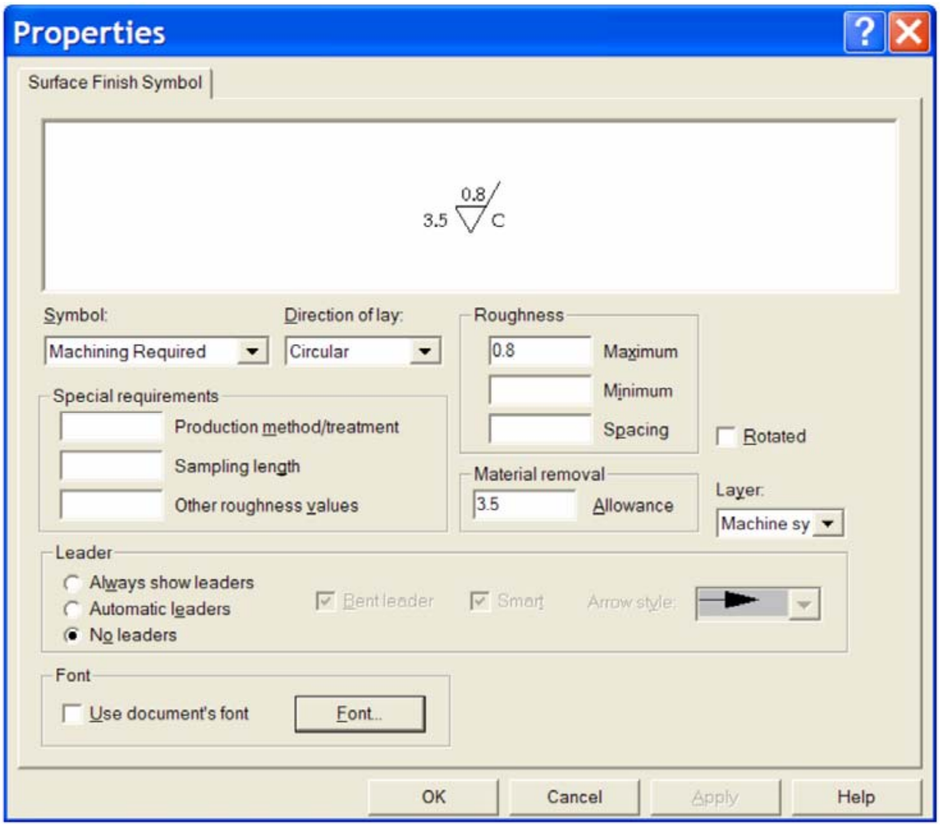


Figure 11

16. We are now ready to place the symbols in the drawing. Move the cursor into the drawing/graphics area of the screen. **DO NOT CLICK THE MOUSE!**

    Notice that the machine symbol is attached to the cursor. The symbol may now be attached to all surfaces that have to be machined. See Figure 12. However, the Properties Window is covering most of the drawing. The window must be moved out of the way. (Note; DO NOT CLOSE THE WINDOW).

    To move the window place the cursor into the blue bar at the top, hold down the left button and drag the window to the left of the screen. This will uncover the drawing. Most of the window is now '**out of sight**'. Place the symbols, one by one, onto the surfaces indicated in Figure 12. Notice that you may place as many symbols as are needed. When all symbols are in place we may close the Properties Window to end the command.

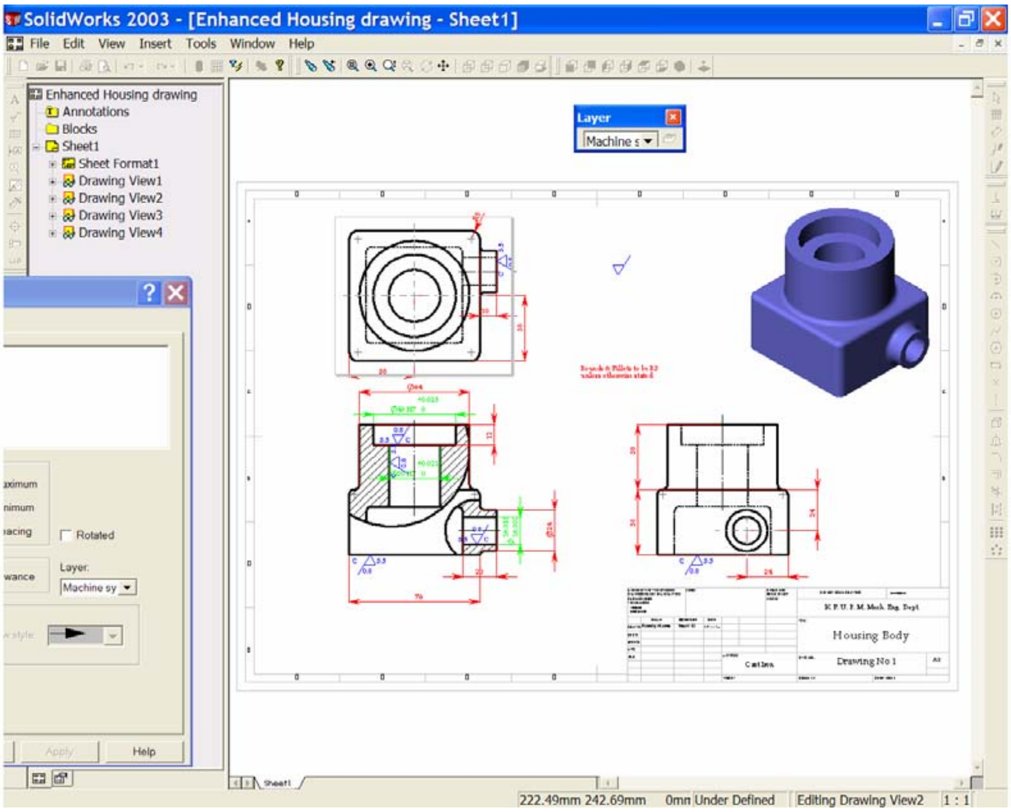


Figure 12

17. The dimensions and machine symbols may need to be moved a little if they overlap each other or if they are too close together. We may drag the dimensions or the symbols to better locations within the views by selecting them with the left button of the mouse and holding the button down as we drag. When you are finished your drawing should look similar to Figure 13.

SAVE YOUR WORK!!

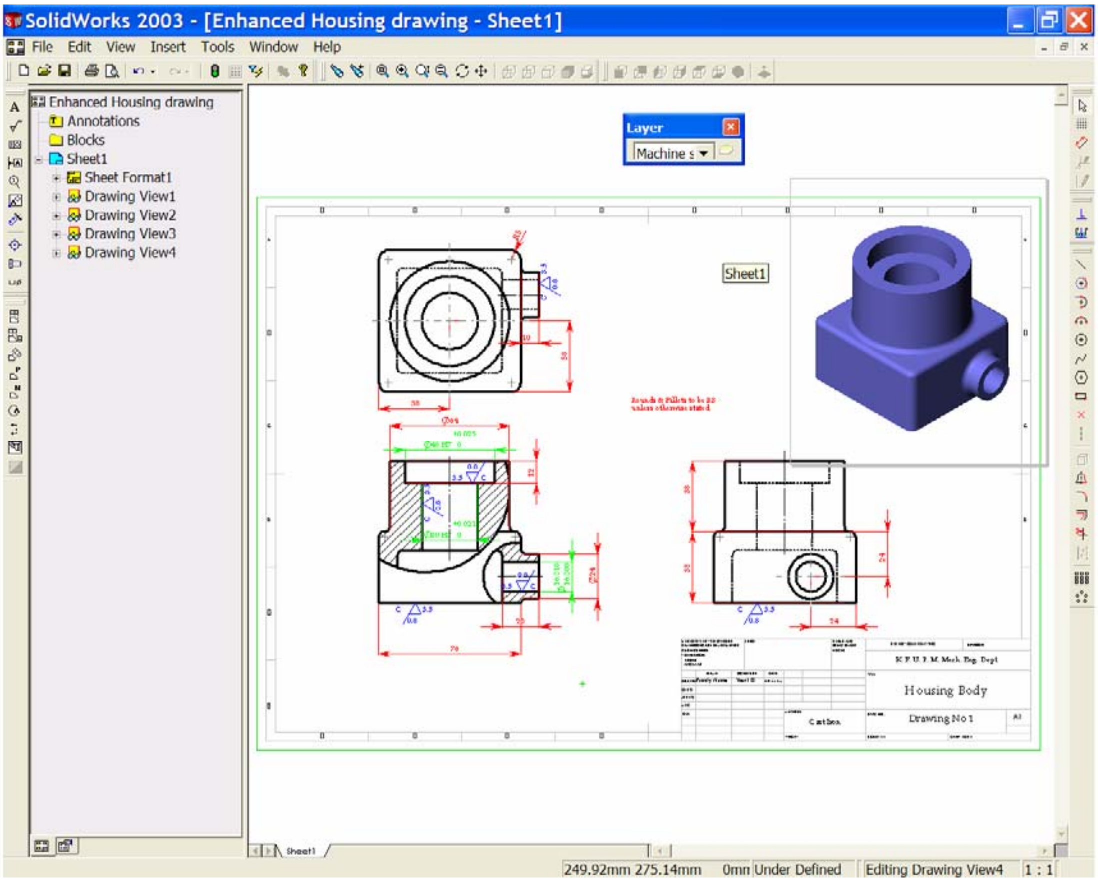

SolidWorks 2003 - [Enhanced Housing drawing - Sheet1]
File Edit View Insert Tools Window Help
Enhanced Housing drawing
Annotations
Blocks
Sheet1
Sheet Format1
Drawing View1
Drawing View2
Drawing View3
Drawing View4
Layer
Sheet1
Housing Body
Drawing No 1
249.92mm 275.14mm 0mm Under Defined Editing Drawing View4 1 : 1

Figure 13

Module 7: Construct Bushes and Produce Detail Drawings

Create models and detail drawing for the bushes given in Figures 1 and 2.

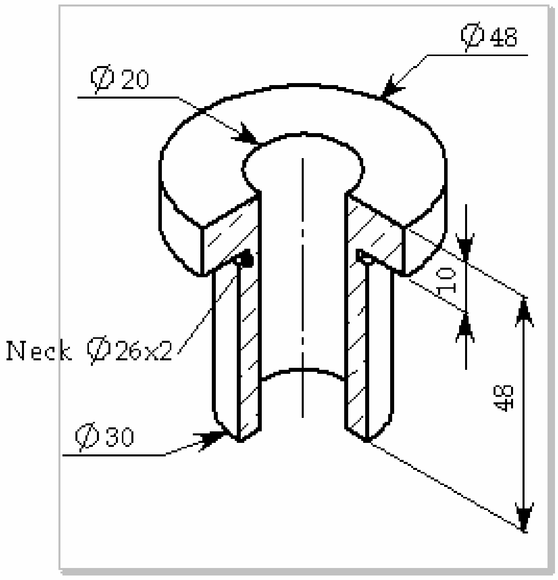


Figure 1 Bushing

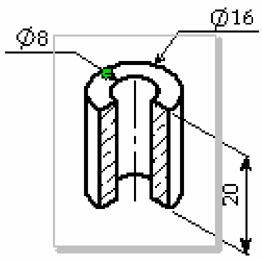


Figure 2 Plain Bush

Module 8: Assembling Housing Parts

1. Open the Housing and the small bush.

2. From '**New**' open an Assembly File  and select '**Tile Horizontally**'. See Figures 1 & 2.

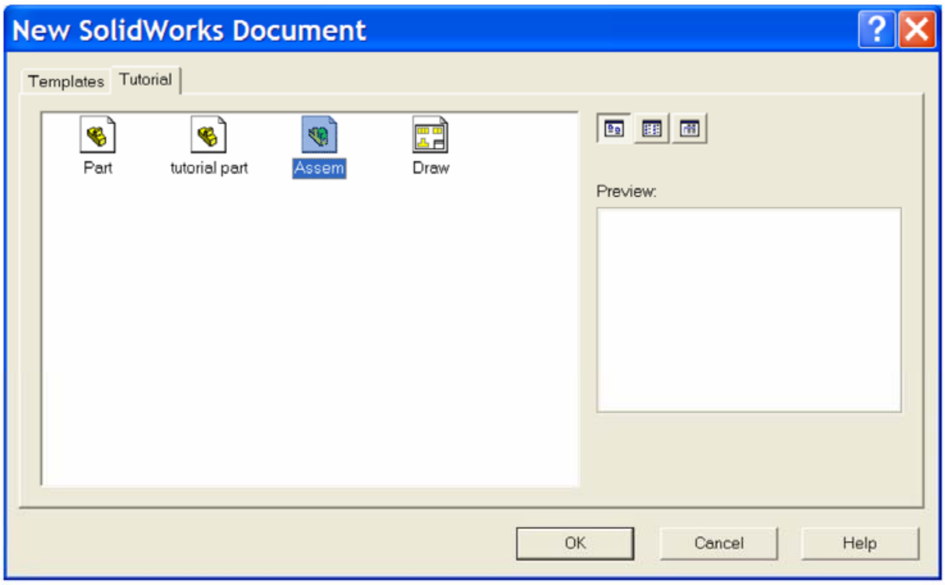


Figure 1

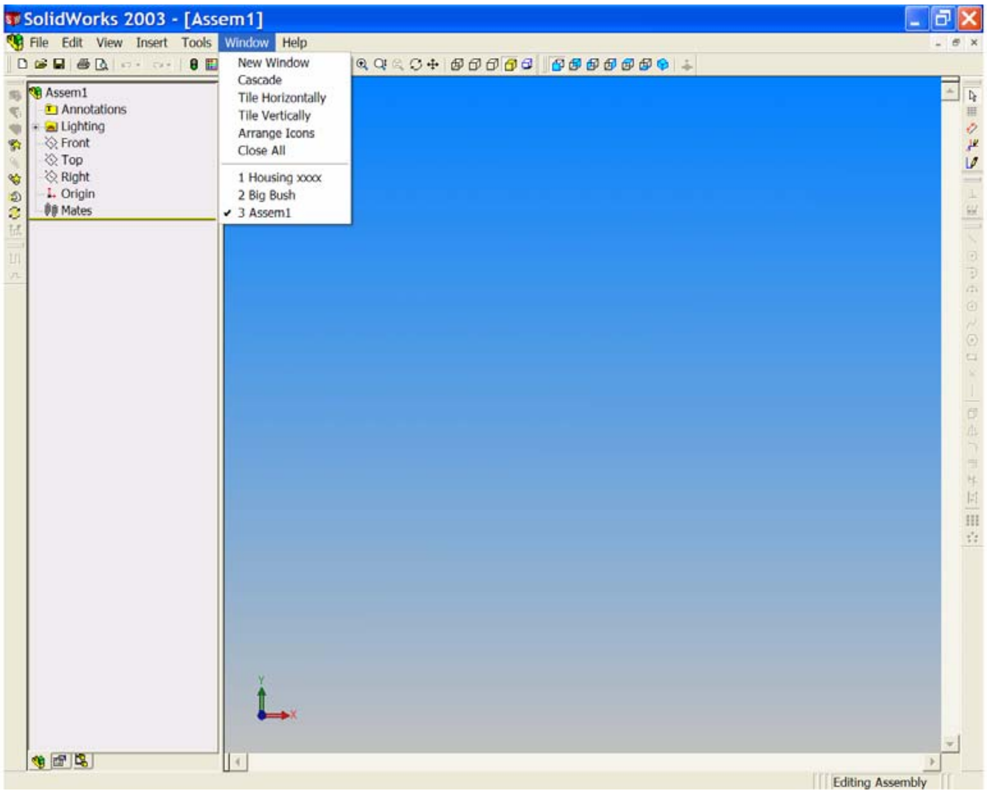


Figure 2

Notice the 'Assembly Design Tree' and the 'Assembly Toolbar'. In Figure 3, the screen displays all three windows.

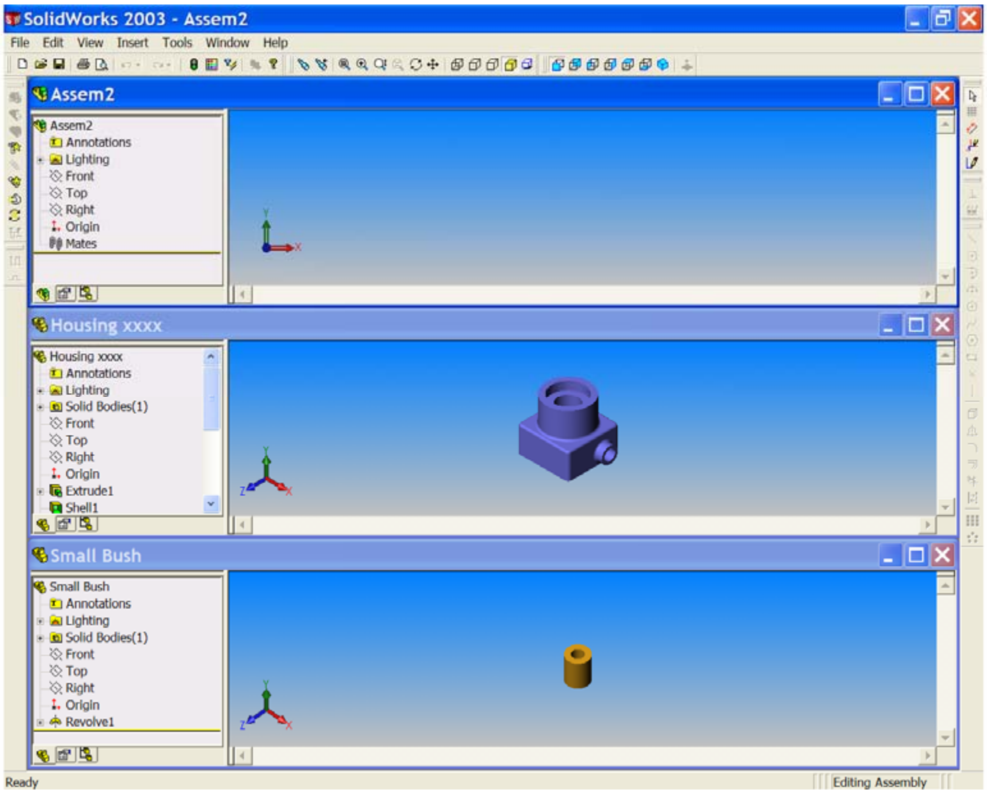


Figure 3

3. Drag the '**Housing icon**' from the top of the **HOUSING FEATURE MANAGER TREE** in the Housing part window, and drop it in the **FEATURE MANAGER DESIGN TREE OF THE ASSEMBLY WINDOW**. See Figure 4.

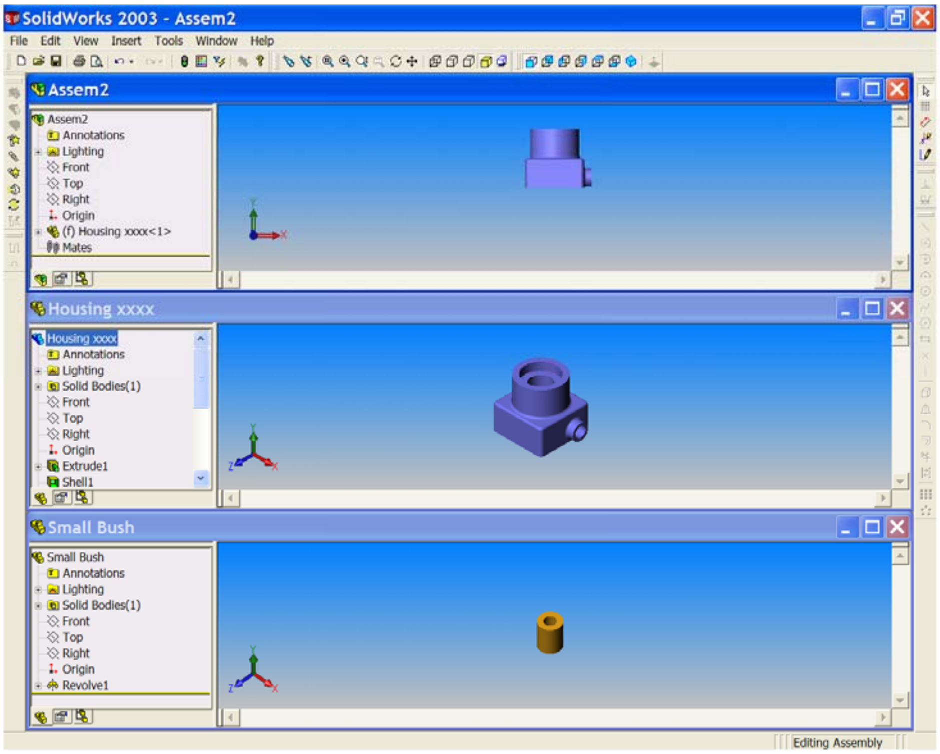


Figure 4

Notice the shape of the cursor as you move into the Feature Manager Design tree of the assembly. When the main part of an assembly is added this way, the part is fixed at the assembly file origin.

- *The origin of the part coincides with the origin of the assembly origin.*
- *The part and assembly planes are aligned.*

4. Drag the '**small bush icon**' from the top of its **FEATURE MANAGER DESIGN TREE** window, and drop it in the **GRAPHICS AREA** of the assembly window close to the Housing. See Figure 5.

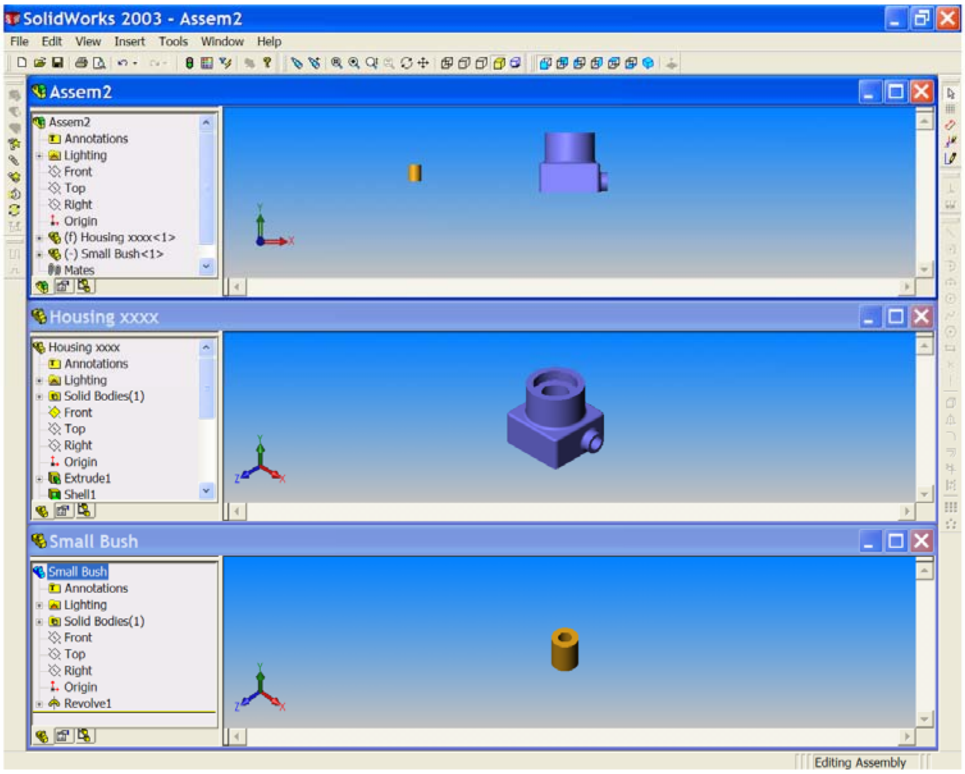


Figure 5

Notice the cursor symbol as the small bush is dropped in the assembly graphics area.

5. **Save as** 'Housing Assembly xxx'. See Figure 6

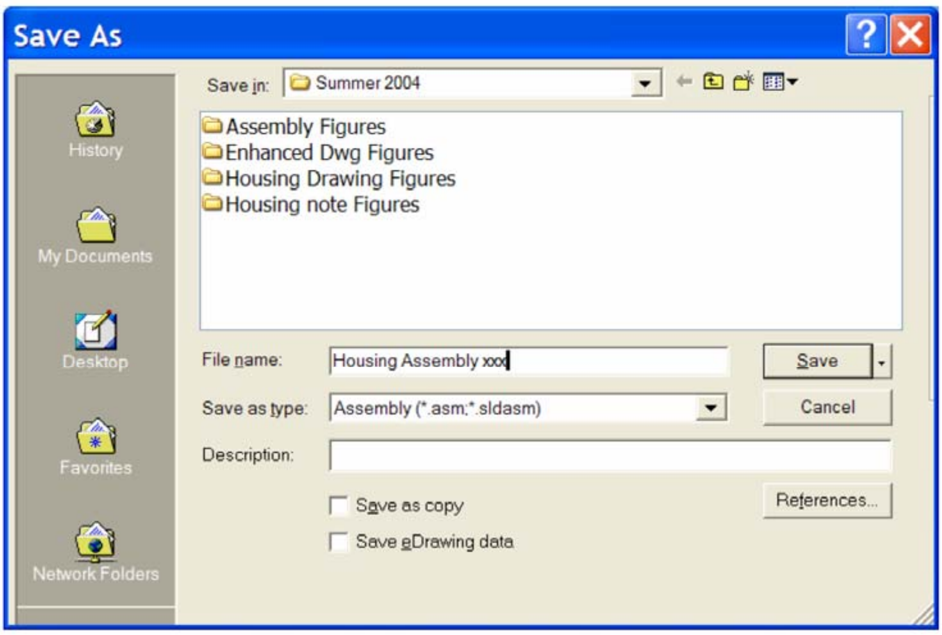


Figure 6

Notice that the file extension is now (.sldasm) indicating an assembly file. We no longer need the part files, but it is useful to keep them open.*

6. Maximise the assembly window and '**Zoom to Fit**'. See Figure 7. We must now create '**Mate Relations**' between these two parts, first to align them and then to fit them together. Click '**Isometric**' then click '**Mate'** from the Assembly Toolbar. See Figure 8.

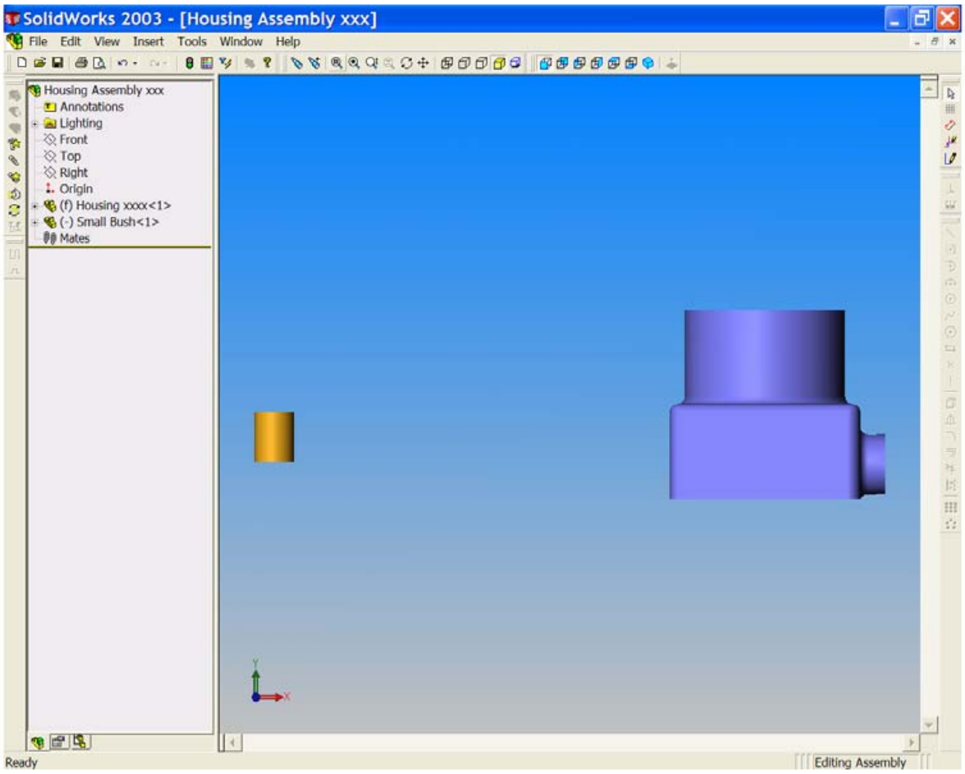


Figure 7

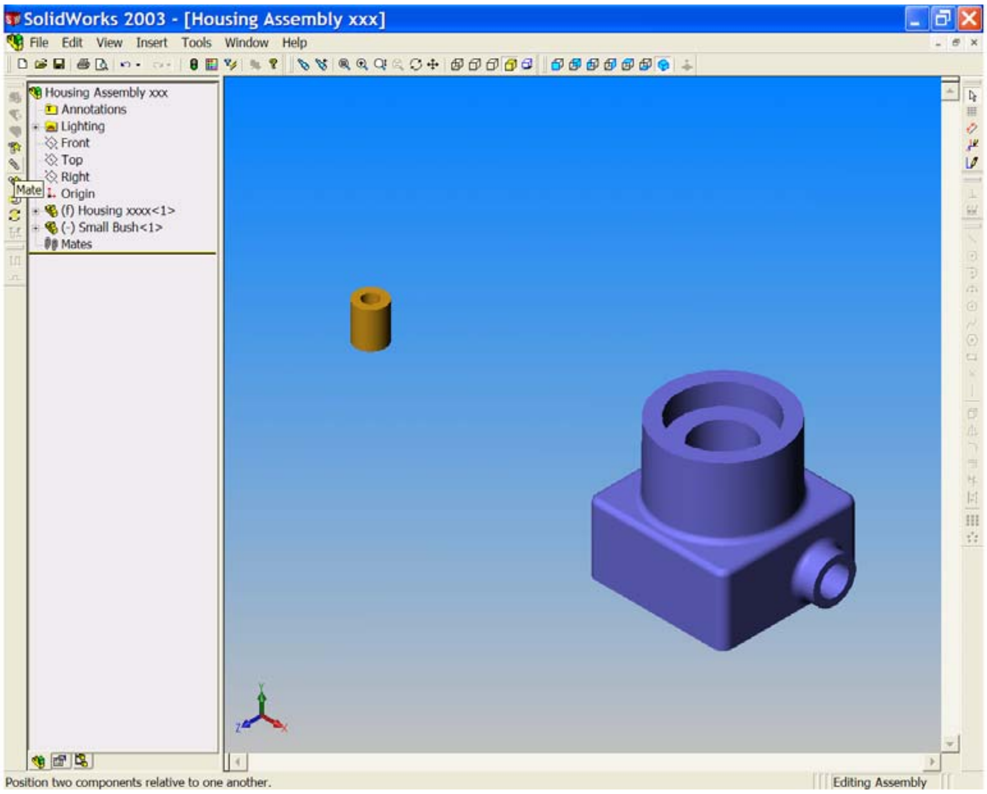


Figure 8

The 'Mate Property Manager' opens. Notice that the 'Entities to Mate' window is active (pink,) See Figure 9. We must now select the surfaces or edges that are to be 'mated'.

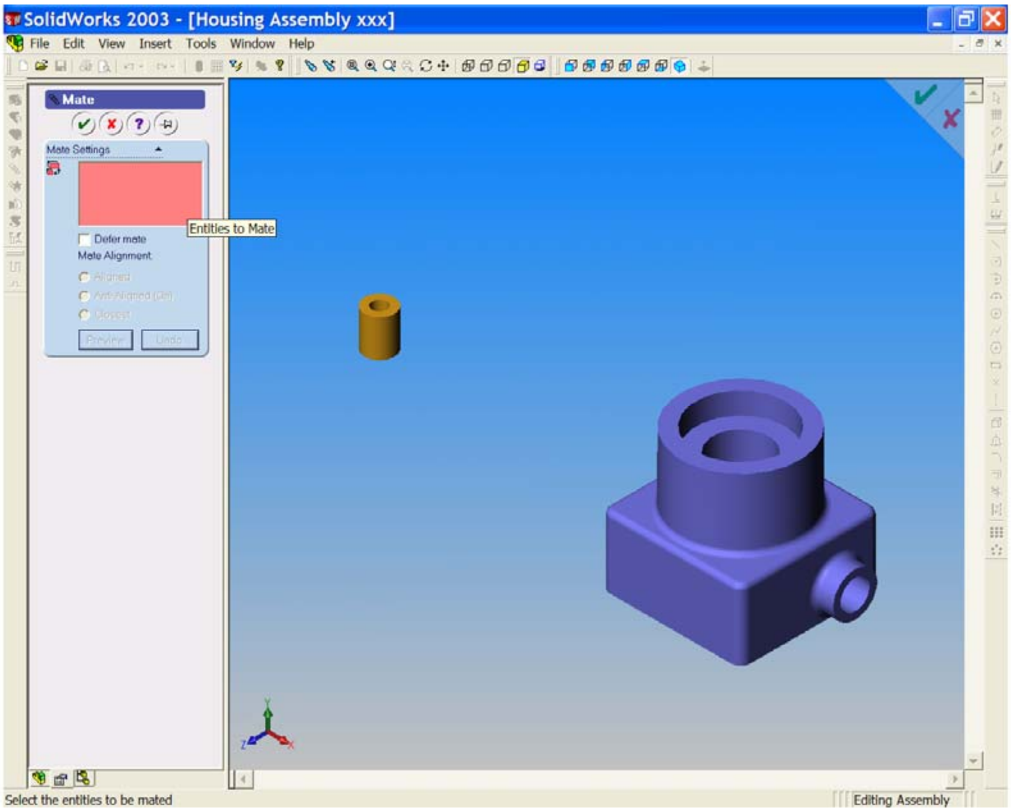


Figure 9

7. Place the cursor on the outer cylindrical surface of the small bush, and left click to select it. The surface is highlighted, the cursor flag is visible and the selected surface is added to the '**Entities to mate**' window. Now place the cursor on the small bore at the right side of the housing. See Figures 10 & 11.

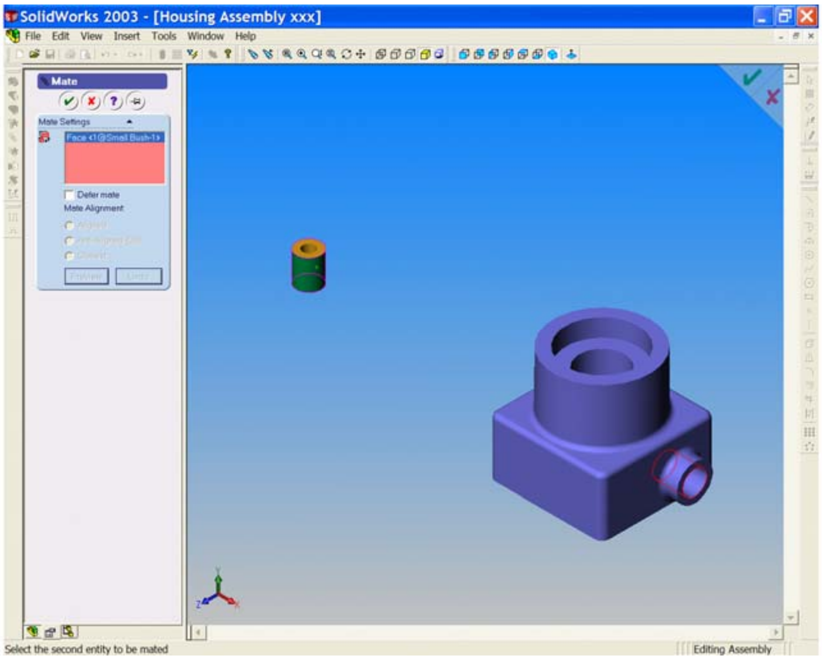


Figure 10

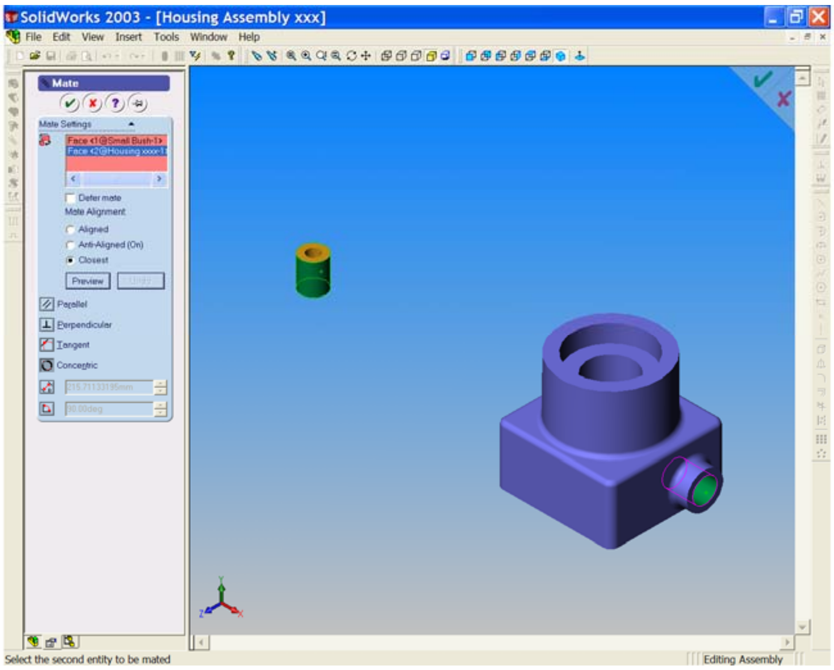


Figure 11

Select the bore surface as before. *Notice that this second surface is added to the 'Entities to mate' window. All possible mate settings are now available.*

Choose ‘**Concentric**’ as the mate type and ensure that ‘**closest**’ is selected as ‘**Mate Alignment**’. Click ‘**Preview**’ and watch the small bore align itself with the centre of the bore. See Figure 13.

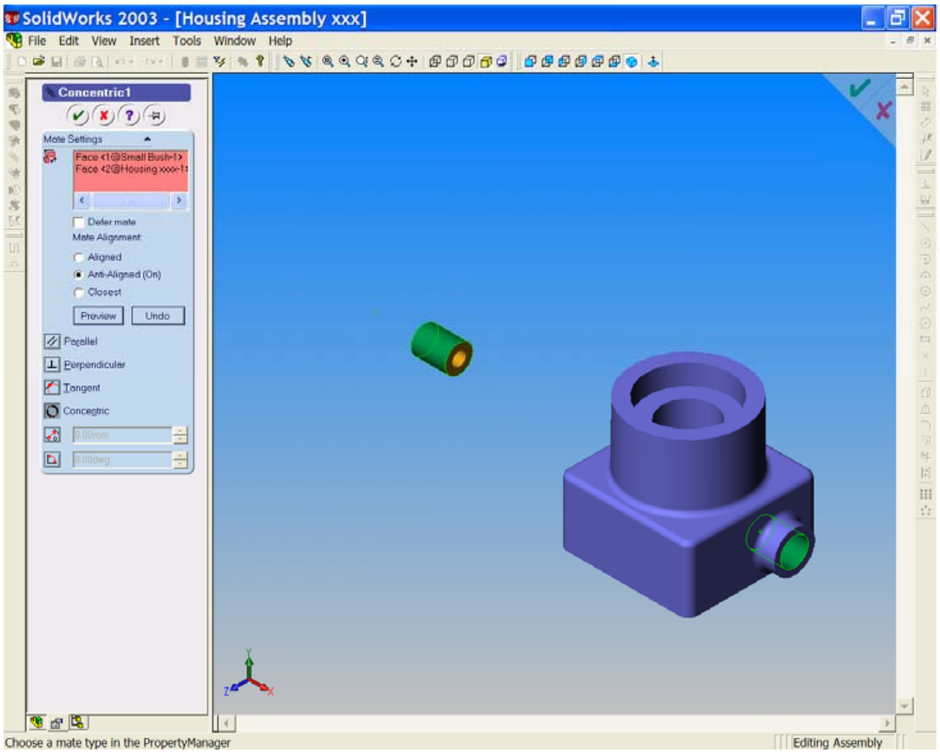


Figure 12

Click **OK** to accept the mate.

8. *The position of the small bush is not fully defined, as indicated by the (-) sign in the Feature Manager Design tree. We must now lock the small bush in its final position.*

Select ‘**Mate**’ once again; select the flat face at the right end of the bush. Now, select the flat face at the right end of the small boss. Both of these faces are highlighted and the mate choices are available. See Figures 13 & 14.

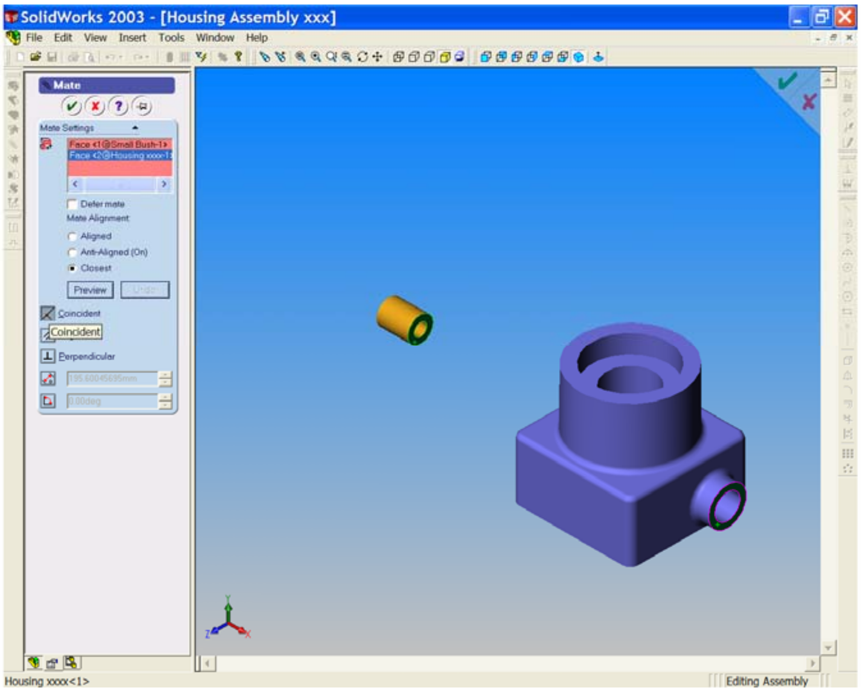


Figure 13

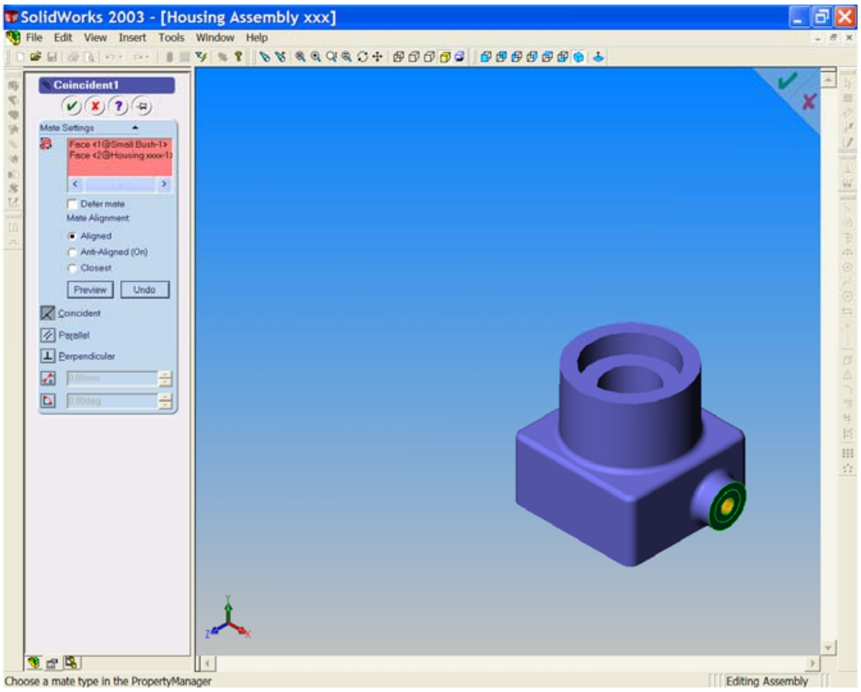


Figure 14

Choose '**Coincident**' and watch as the bush moves into its final position. Click **OK** to end the command. See Figure 15.

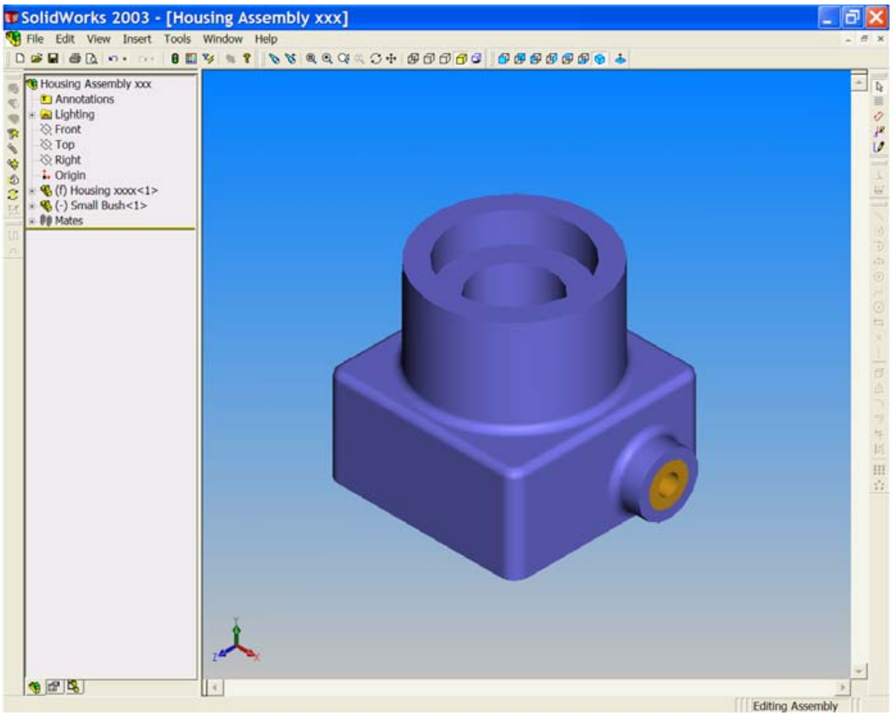


Figure 15

Save your work!!

9. *We must now place the large bush in position*.

From the '**Insert**' drop down menu select '**Component**' & '**From File**'. See Figure 16.

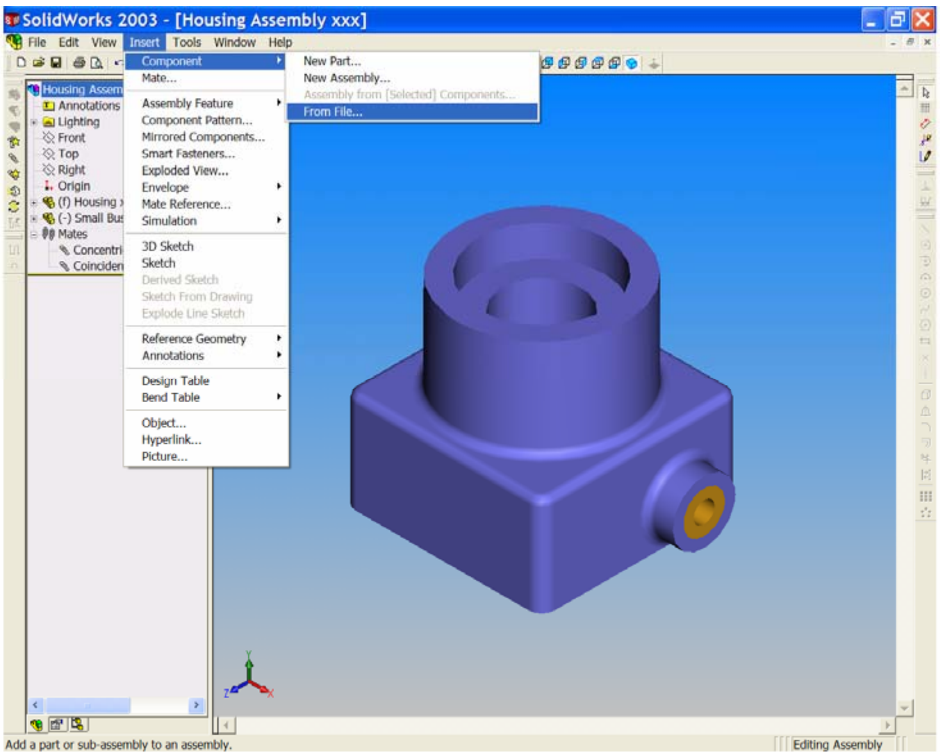


Figure 16

Drop the big bush in the graphics area of the screen.

Note the cursor symbol as you do this.

Now, on your own, mate the big bush to the counterbore of the Housing. When you are finished, your assembly should look similar to Figure 17.

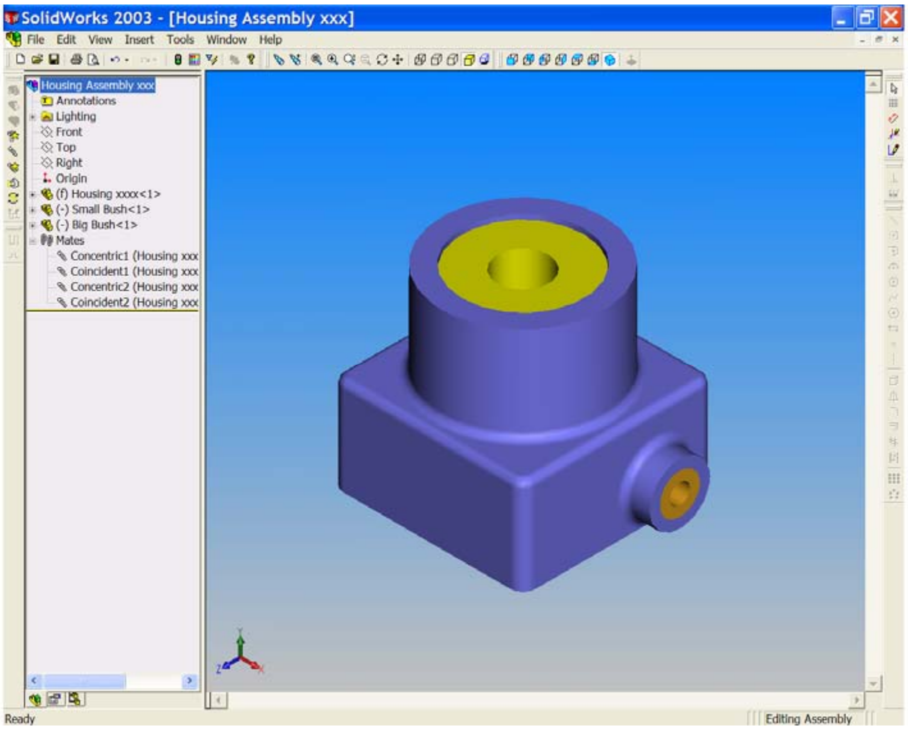


Figure 17

Notice that both bushes still have a (-) sign in the design tree. Why should this be? Which degree of freedom is still not constrained?

10. Save your work  **!!**

Module 9: Assembly Drawing with Bill of Materials

1. Open the A3 Template Drawing Border

2. Save as ‘**Housing Assembly xxx**’.

3. Insert the 3 Standard Views from the Housing Assembly Model File.

4. Add the Broken out sections indicated in Figure 1.

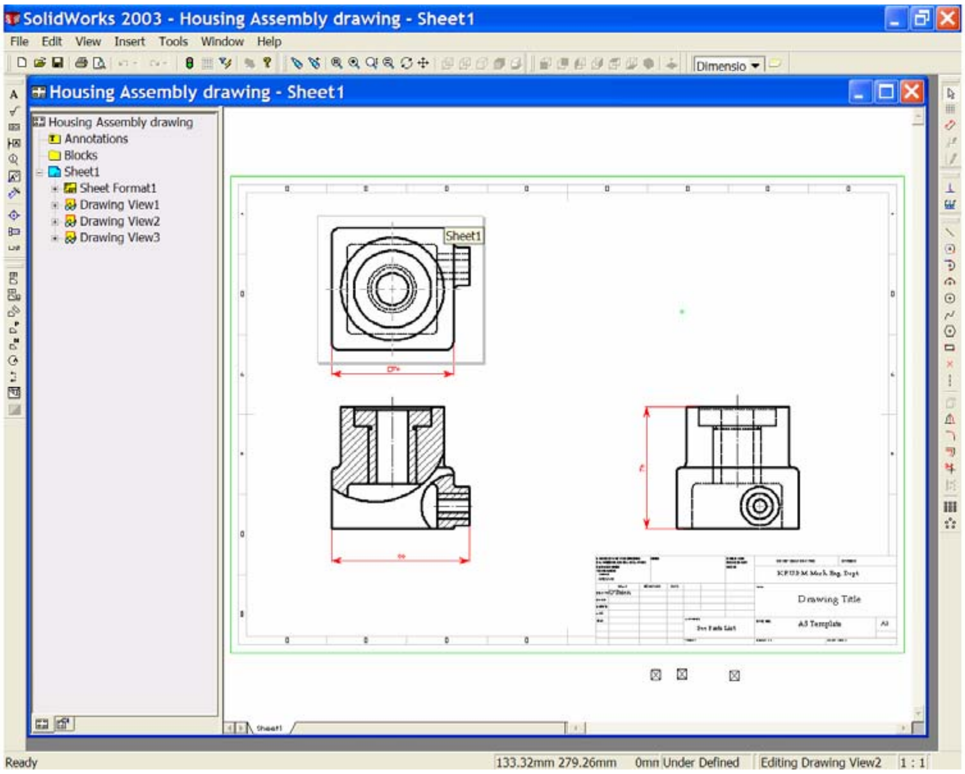


Figure 1

5. Create a layer and use the ‘**dimension tool**'  to add the dimensions shown above.

6. We must now edit the cross hatch pattern for the bushes. Place the cursor in the cross hatch area of the big bush, ‘**right click**’ and select ‘**Properties**’. In the ‘**Area Hatch/Fill**’ Properties box make the changes shown in Figure 2. Choose ‘**Apply**’ to close the window.

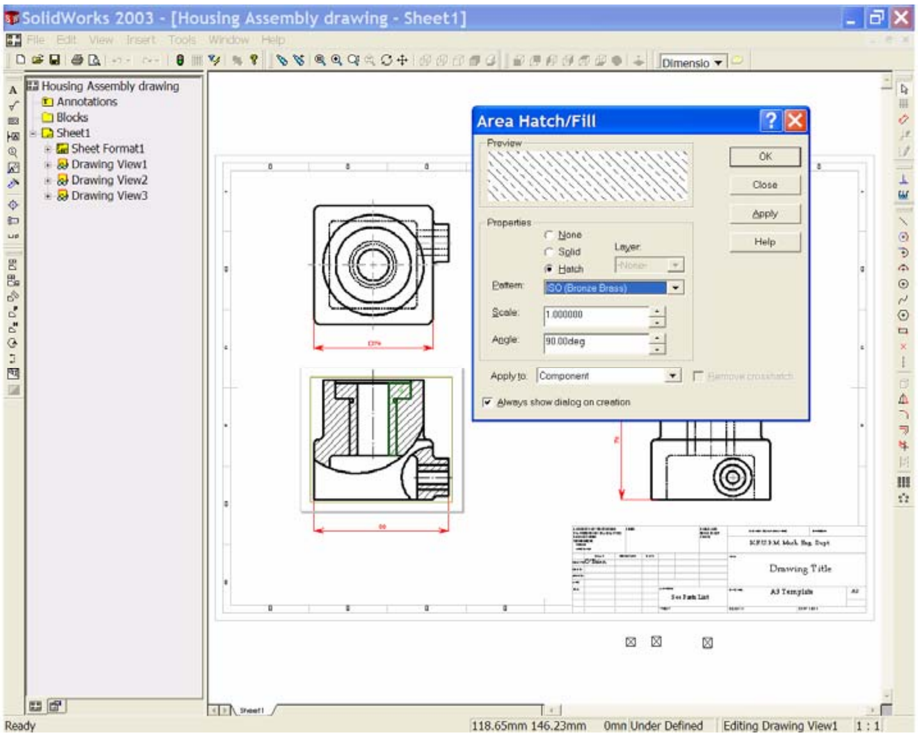


Figure 2

7. Repeat the above procedure for the small bush.

8. **Rebuild**  the drawing and **save** !

9. We are now ready to build and insert the '**Bill of Materials'**. From the '**Tools**' dropdown menu navigate to '**Document Properties**' and '**Detailing**'. Make sure that '**Automatic Update of Bill of Materials**' checkbox is selected. See Figure 3.

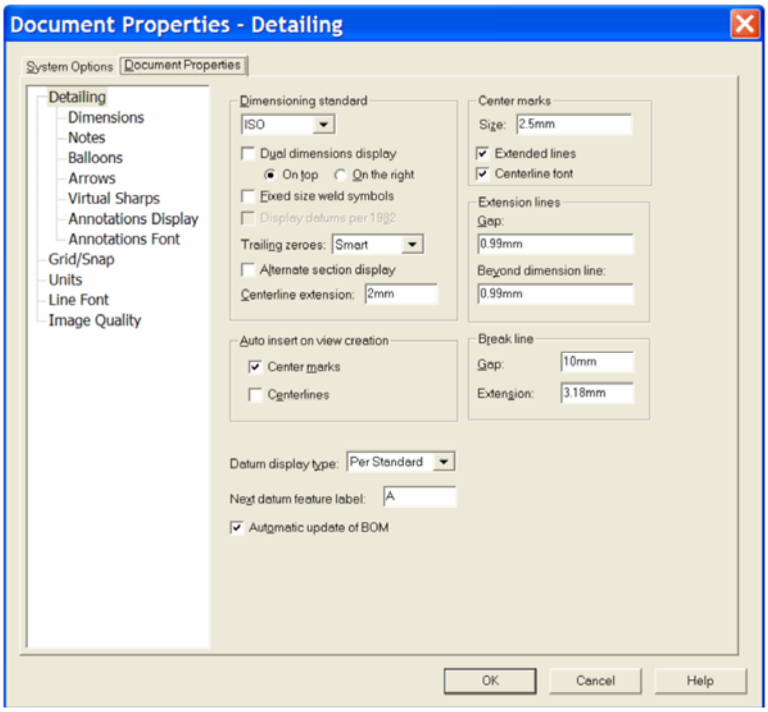


Figure 3

10. Go to '**Balloons**'; in the '**Single Balloon**' area style window, select '**Circular Split line**'. In the '**Balloon Text**' area set '**Upper:**' to '**Item Number**' and '**Lower:**' to '**Quantity**'. In the '**Bent Leaders**' area check the '**Use Bent Leaders**' box. See Figure 4.

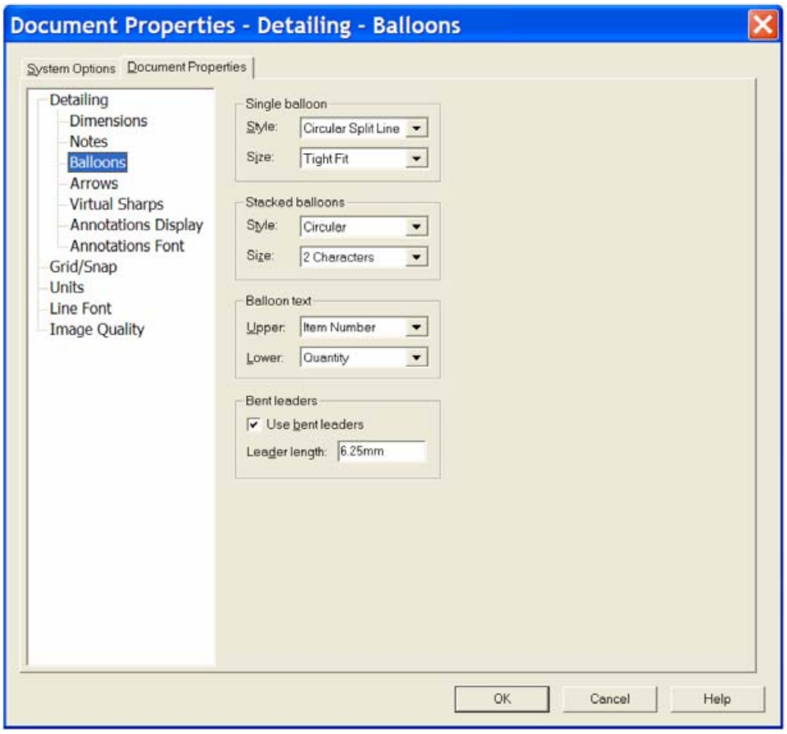


Figure 4

11. Go to '**Annotations Display**' and make sure that '**Display all types**' is checked. See Figure 5.

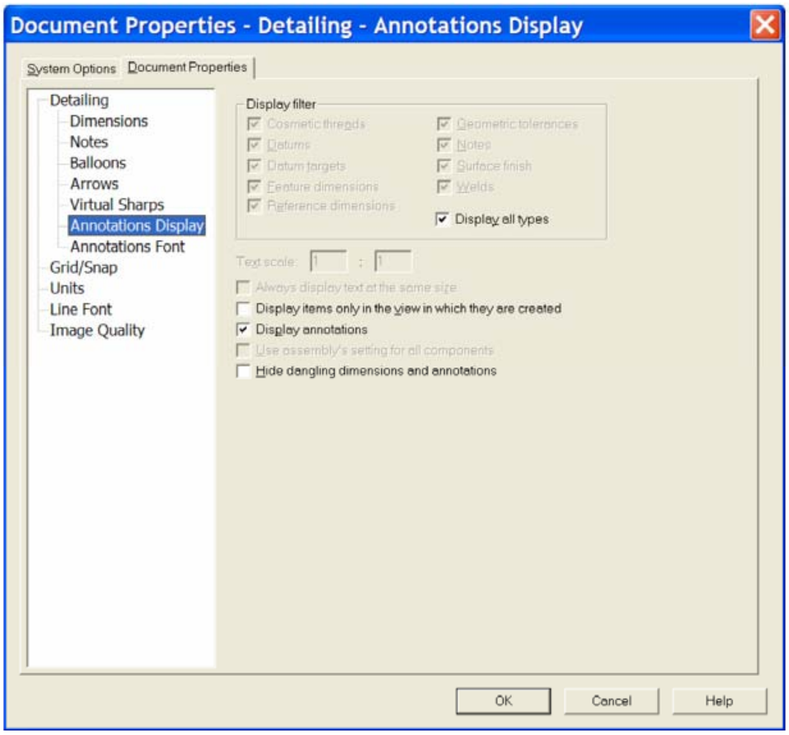


Figure 5

12. Left click **OK** to close the '**Options**' Dialog box.

13. Left click to select the '**Elevation**', and from the '**Insert**' drop down menu choose '**Bill of Materials**'. See Figure 6.

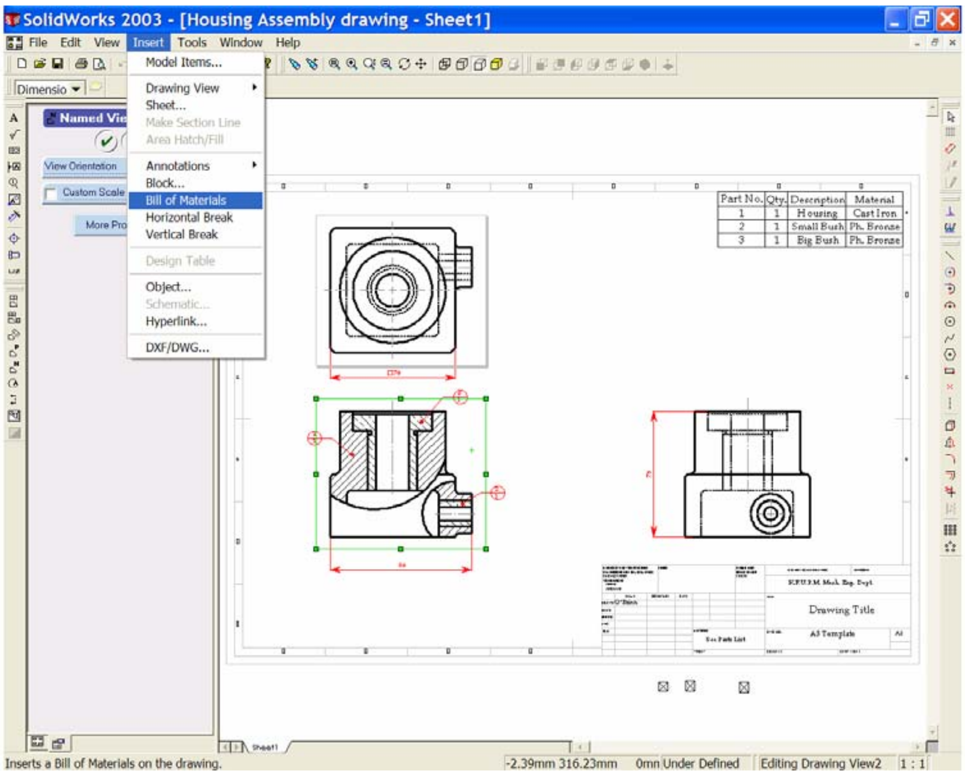


Figure 6

14. From the select ‘**Template Window**’ accept the default template, ‘**bomtemp.xls**'. The ‘**Bill of Materials Properties Window**’ opens. See Figure 7.

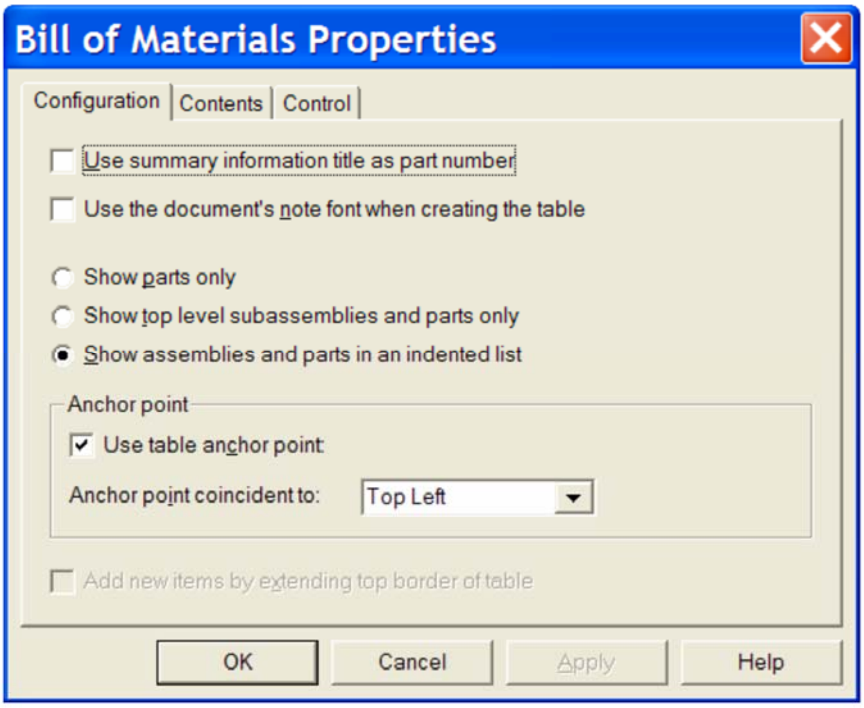


Figure 7

15. Make sure that the following settings are in place:

- Clear the ‘Use documents note font when creating the table’.
- Choose ‘Show assemblies and parts in an indented list’.
- In the Anchor point area check ‘Use table anchor point’.
- Set Anchor point coincident to ‘Top Left’.

Now click **OK** to compile and display the ‘Bill of Materials’.

16. Right click in the ‘**Bill of Materials**’ and select ‘**Anchor, Top Right**’. See Figure 8.

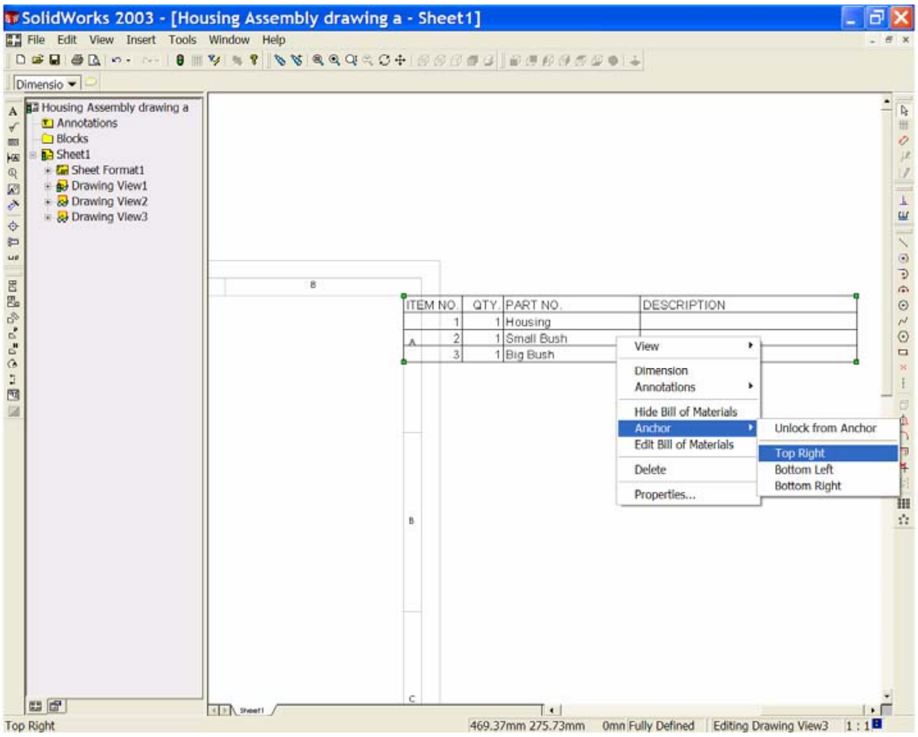


Figure 8

*Notice that the '**Bill of Materials**' is attached to the drawing border at the Top Right corner. By changing the '**Bill of Materials**' anchor point to Top Right these two corners will be coincident.*

17. Use the '**Zoom to Area**' tool  and zoom to enlarge the Bill of Materials. Right click in the bill of materials and choose '**Edit Bill of Materials**'. When the Bill of materials is active, it is displayed with a shaded border, with row and column headers and the **Excell** toolbars replace the **SolidWorks** toolbars. See Figures 9 & 10.

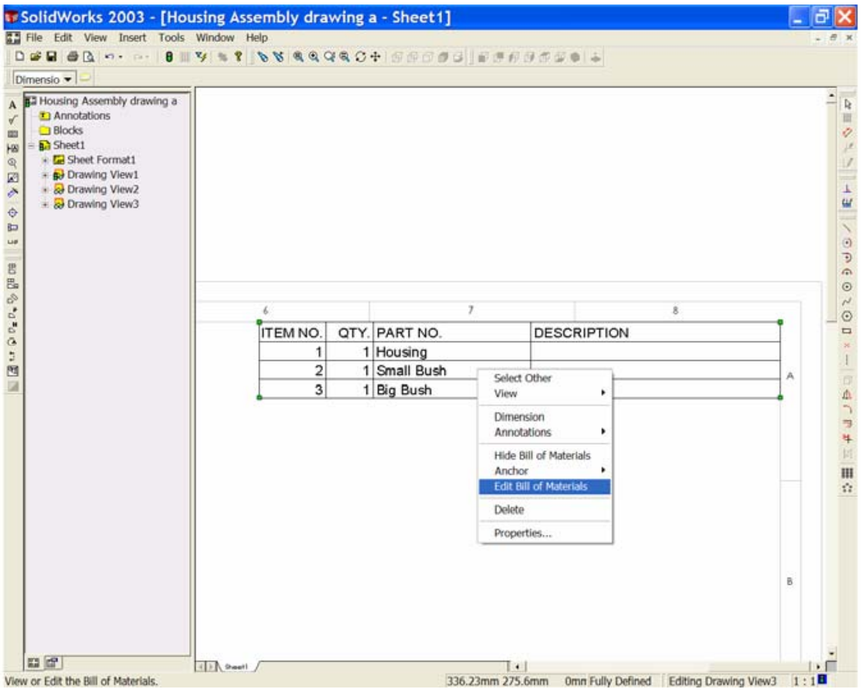


Figure 9

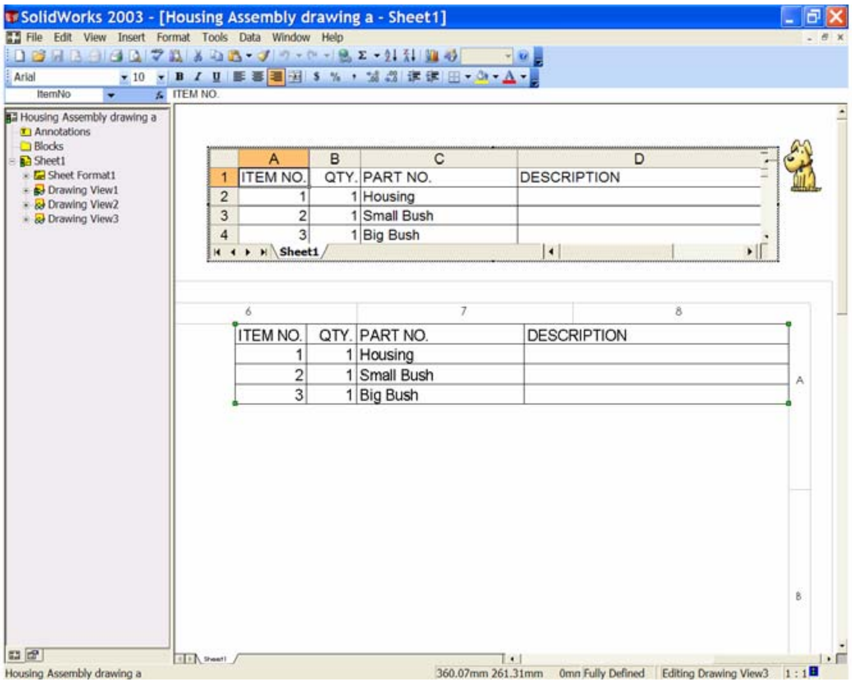


Figure 10

18. We wish to edit the contents.

- In cell **A1** delete '**Item No.**' and replace it with '**Part No.**'.
- In cell **C1** delete '**Part No.**' and replace it with '**Description**'.
- In cell **D1** delete '**Description**' and replace it with '**Material**'.
- In cell **D2** enter 'Cast Iron'.
- In cells **D3&D4** enter '**Ph. Bronze**'.

See Figure 11.

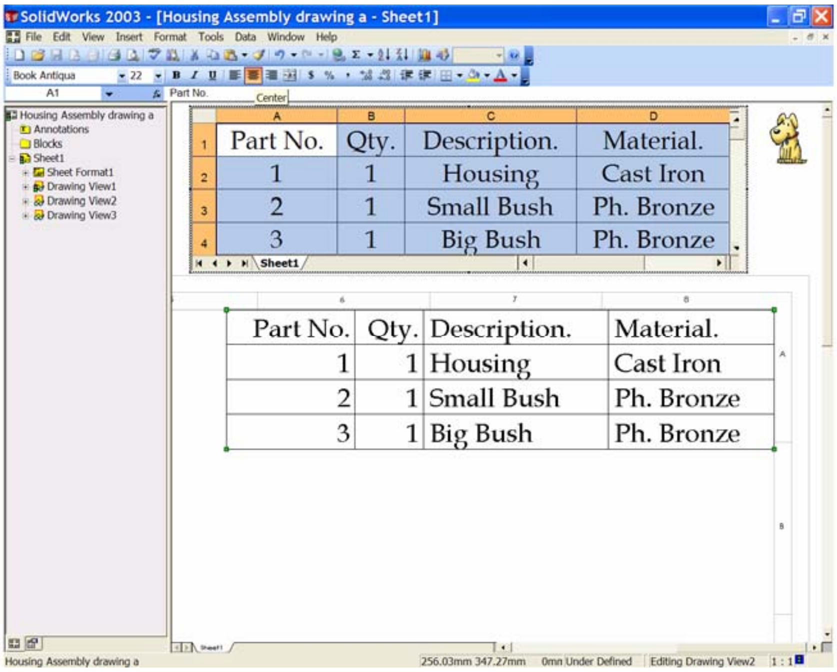


Figure 11

19. We may also change the font, re-size the text and so on. Select '**Book Antiqua**' with a font size of '**22**'. From '**Format**' select '**Row**' and '**AutoFit**'. Then select '**Column**' and '**AutoFit Selection**'.

20. Left click outside the '**Bill of Materials**' to close **Excell** and return to **SolidWorks**, with the updated data. See Figure 12.

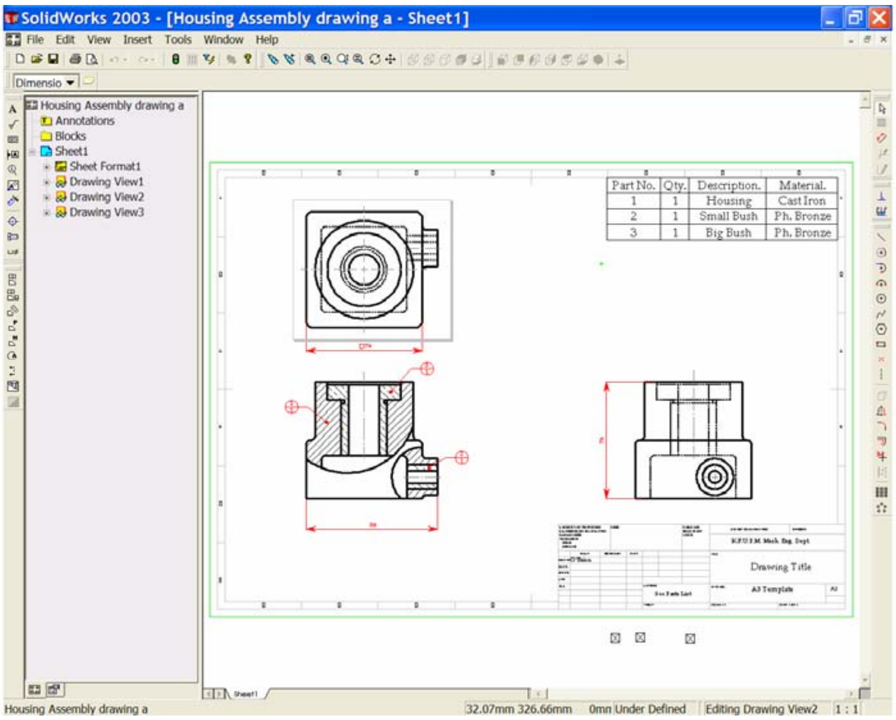


Figure 12

Save your work!

21. We now add ‘**Balloon References**’ to the drawing; click ‘**Balloon**’ on the ‘**Annotations Toolbar**’. The ‘**Balloon Property Manager**’ opens. See Figure 13.

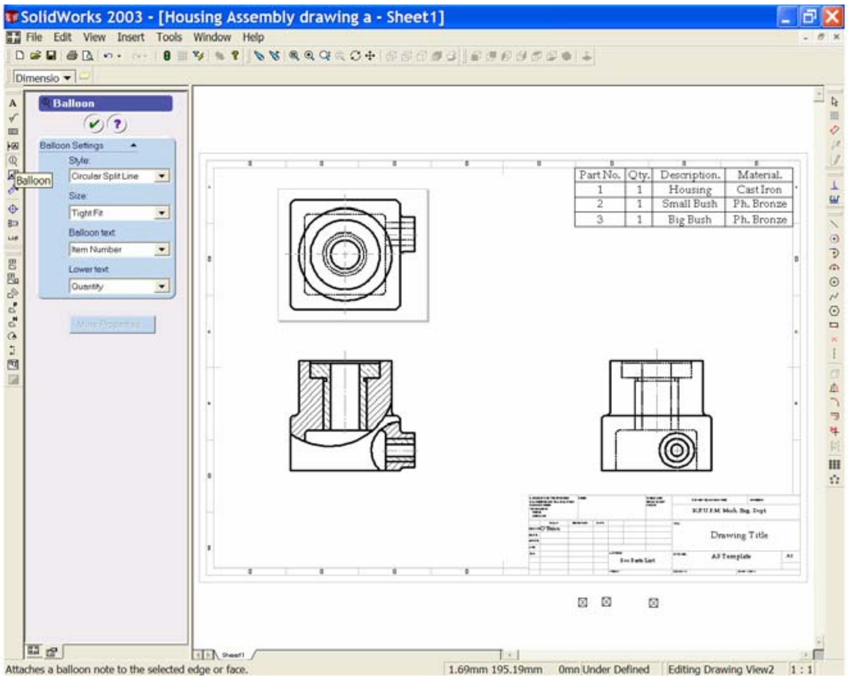


Figure 13

22. Left click a component in the drawing view.

A balloon attaches itself to the part. The numbers correspond to the part number and the quantity listed in the Bill of Materials.

Add Balloons to the other two parts then click **OK** . To move the balloon or its leader arrow, select and drag by the green handles. See Figure 14.

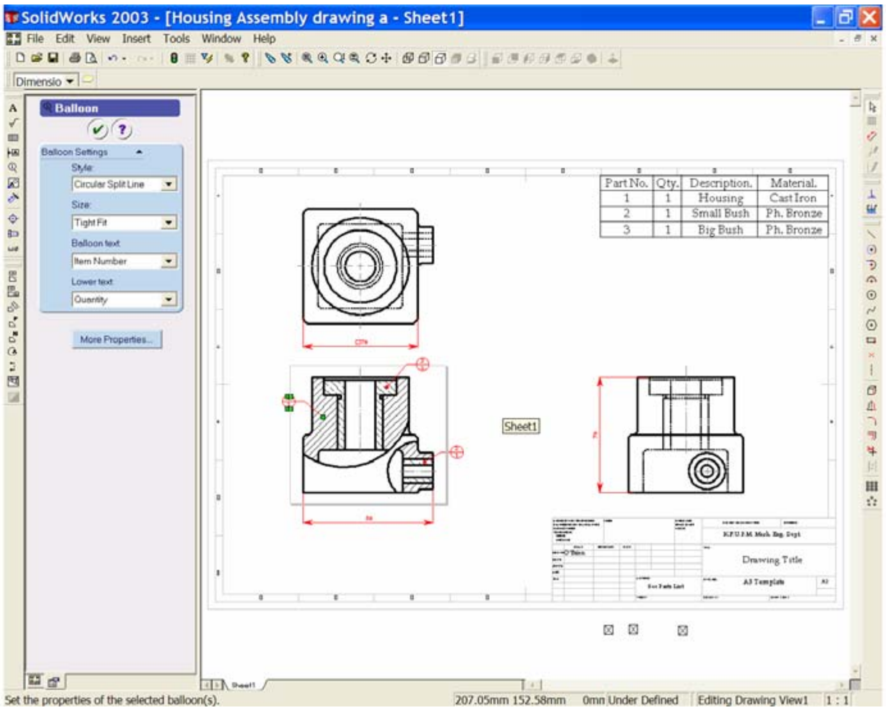

SolidWorks 2003 - [Housing Assembly drawing a - Sheet1]
File Edit View Insert Tools Window Help
Balloon
Balloon Settings
Style:
Circular Split Line
Size:
Tight Fit
Balloon text
Item Number
Lower text
Quantity
More Properties...
Part No. Qty. Description. Material.
1 1 Housing Cast Iron
2 1 Small Bush Ph. Bronze
3 1 Big Bush Ph. Bronze
Sheet1
Drawing Title
Set the properties of the selected balloon(s).
207.05mm 152.58mm 0mm Under Defined Editing Drawing View1 1 : 1

Figure 14

Module 10: Sectional Views

1. Open a ‘**New Part**’ file and set the **units** to inches, decimal and three places. See Figure 1.

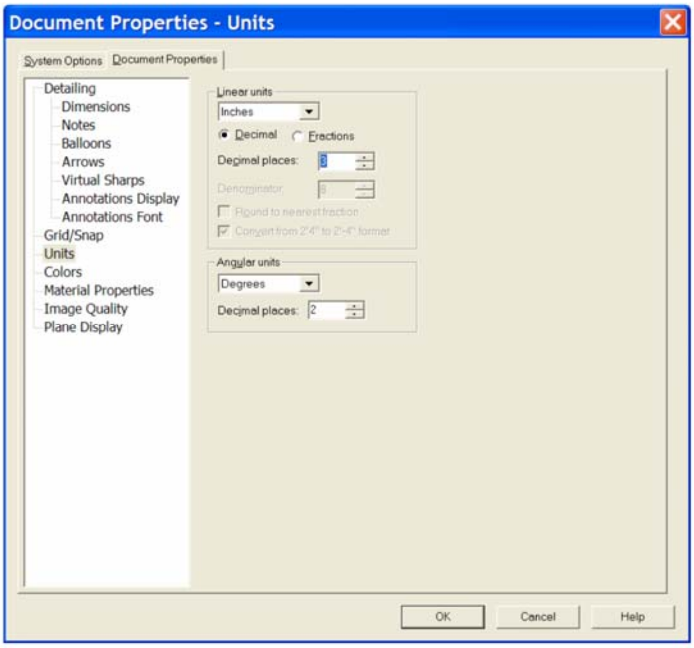


Figure 1

2. Open a **Sketch** on the **Top Plane** and sketch the profile shown in Figure 2.

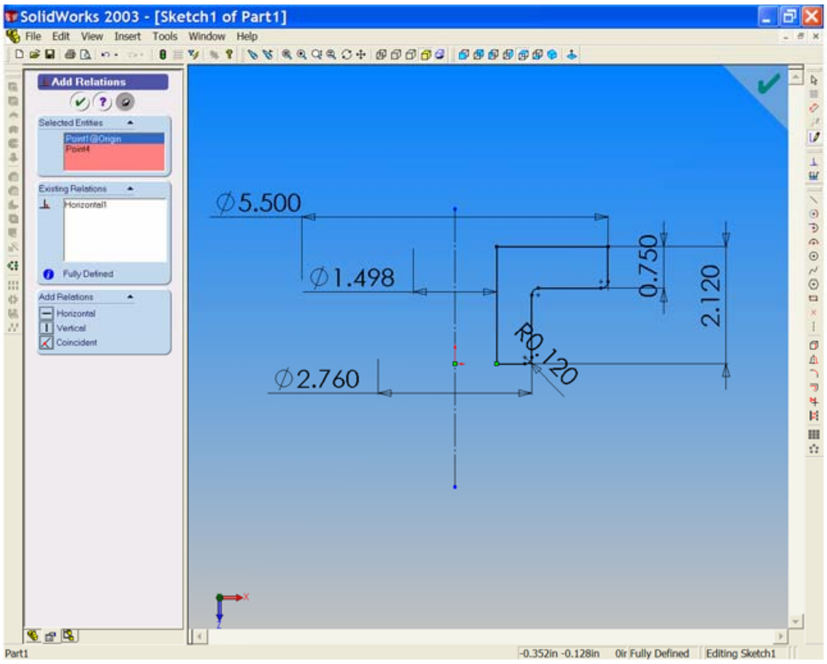


Figure 2

3. Select the ‘**Revolved Boss/Base’** tool  to create a revolved model from the sketch.

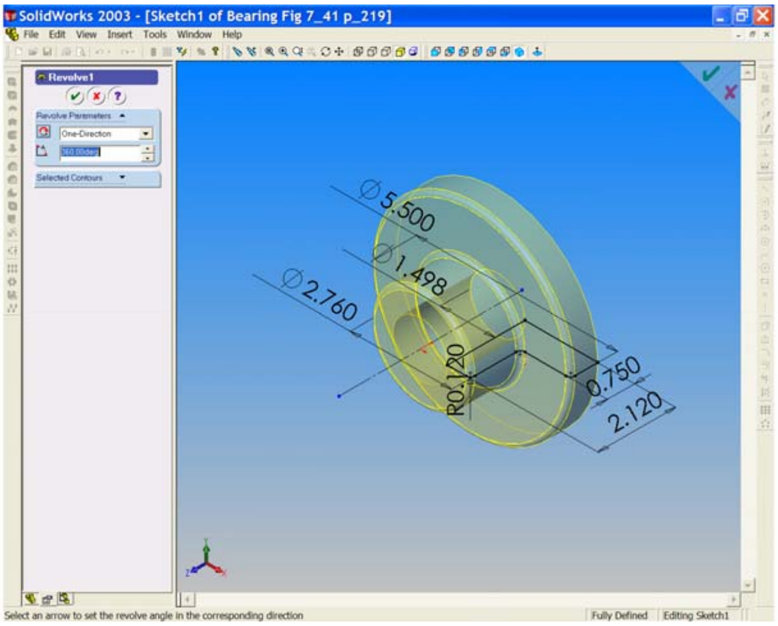


Figure 3

Accept the default revolve type and angle settings, click OK to generate the solid model.

4. **Save as** 'Bearing Fig 7_41'.

5. We must now place the 4 bolt holes in the flange. Select the front face of the flange and open a sketch on this surface. Select '**Normal To**' and draw a circle as shown in Figure 4.

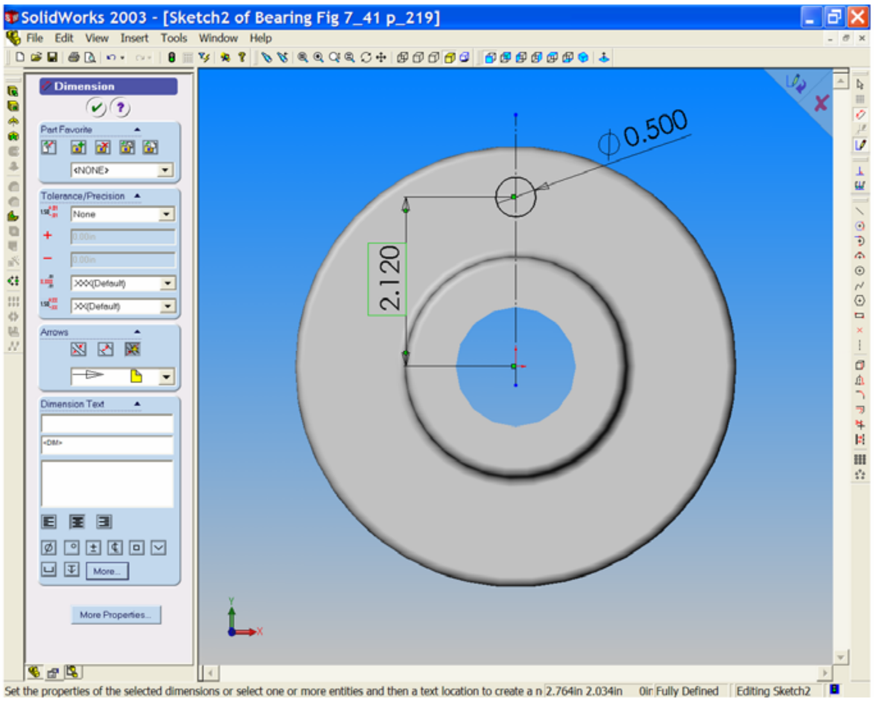


Figure 4

6. Select '**Extruded Cut**' to form the bolt hole '**Through All**'.

7. Turn on ‘**Temporary Axes**’ from the ‘View’ dropdown menu. Select ‘**Circular Pattern**’ , the Circular Pattern Properties Manager opens with ‘Cut-Extrude1 listed in the Features to Pattern field. The ‘Pattern Axis’ field is pink; select the temporary axis that passes through the flange bore. Axis <1> is now listed and a preview of the patterned features is shown on screen. Increase the number of instances from 2 to 4 by using the ‘up arrow’. Notice the preview changing as the value changes. Select OK  See Figure 5.

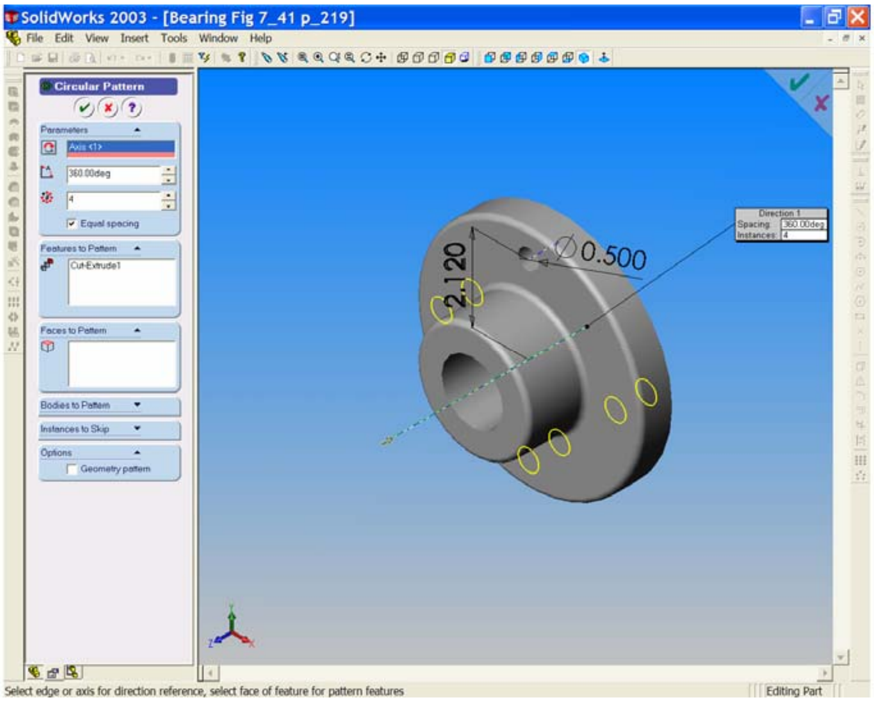


Figure 5

8. The model is complete and we may now turn off the temporary axes.

Save your work!!!

9. Now that the model is complete we must now prepare an Imperial drawing border. Using your previous instructions set up an imperial border using ‘C-Landscape’ as your base template. 

Ensure that:

- Units are set to inches and 3 decimal places.
- Scale is set to 1:1.
- Type of projection is ‘Third Angle’.
- Drawing Title is ‘Bearing’.
- Material is ‘Cast Iron’.
- Finish is ‘See Machine Symbols’
- ‘KFUPM Mech. Eng. Dept.’ is added.
- Your family name, ID No and section number are given
- The date is added.

10. Insert the three orthographic views with tangent edges removed. Position the views carefully within the border. You will have to modify the DWG. NO. (Use ‘Properties’). See Figure 6.

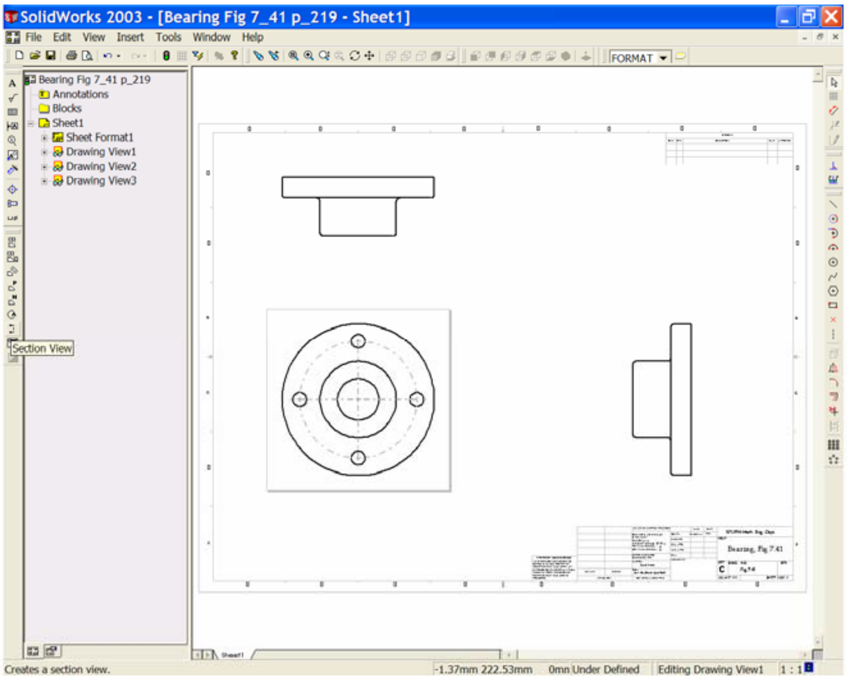


Figure 6

11. We will add a ‘**Full Section**’ to replace the Right End Elevation. Select the ‘**Section View**’ button from the ‘Drawing View’ toolbar. Place the cursor above the vertical centre line of ‘Elevation’ left click once and draw the section line (cutting plane) vertically down through the model to the bottom. *Notice the cursor symbol change as it indicates what you are doing.*

    When you have drawn the line to the bottom of the model, left click once.

    Notice that the cutting plane with viewing direction arrows is added to the Elevation and that the ‘Section View’ appears on screen. The ‘Section View Property Manager’ is open. The cutting plane arrows may be pointing in the wrong direction. If so, we can use the ‘Flip direction’ by checking the box in the property manager.

    Left click to check the box and observe the result on screen. When you have the desired Section View left click once in the graphics area to place the view (with tangent edges removed). See Figure 7.

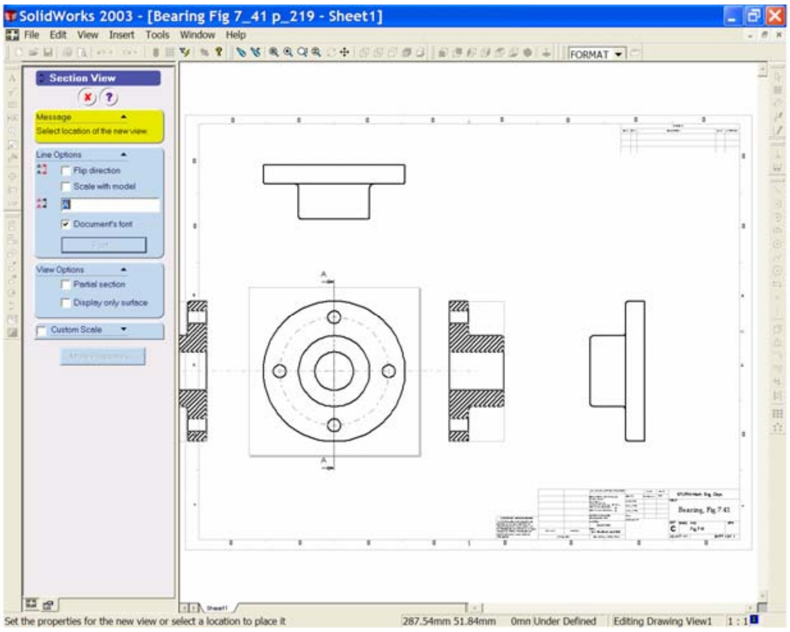


Figure 7

12. The Standard Right End Elevation is not required and may be hidden. Right click inside the border of this view and choose '**Hide View**'. See Figure 8.

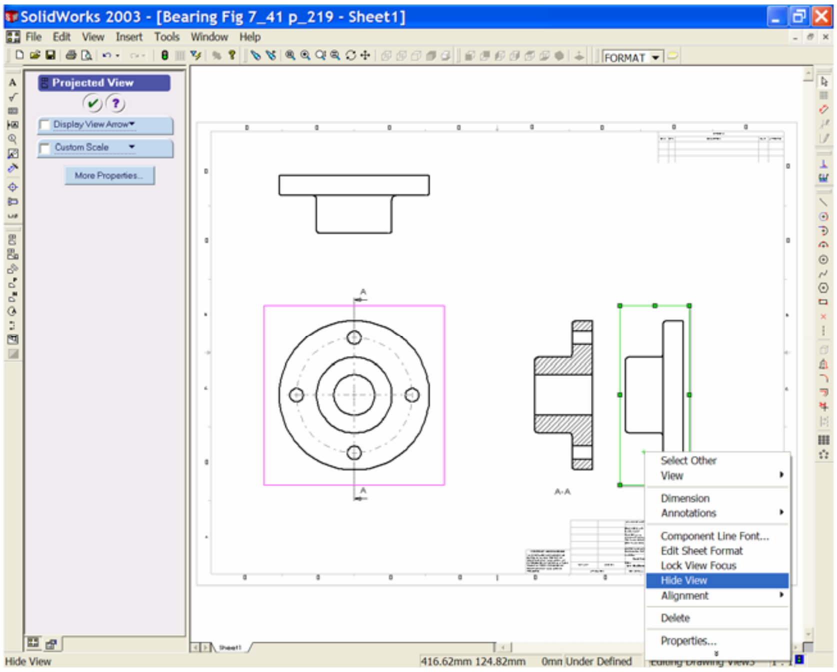


Figure 8

13. Now we need to add centre lines to the views. Activate the section view and select '**Centreline**' from the '**Annotations**' toolbar. Centrelines are placed through the cylindrical parts. Switch on 'Hidden Edges' for the Plan and repeat the procedure for the Plan. See Figure 9.

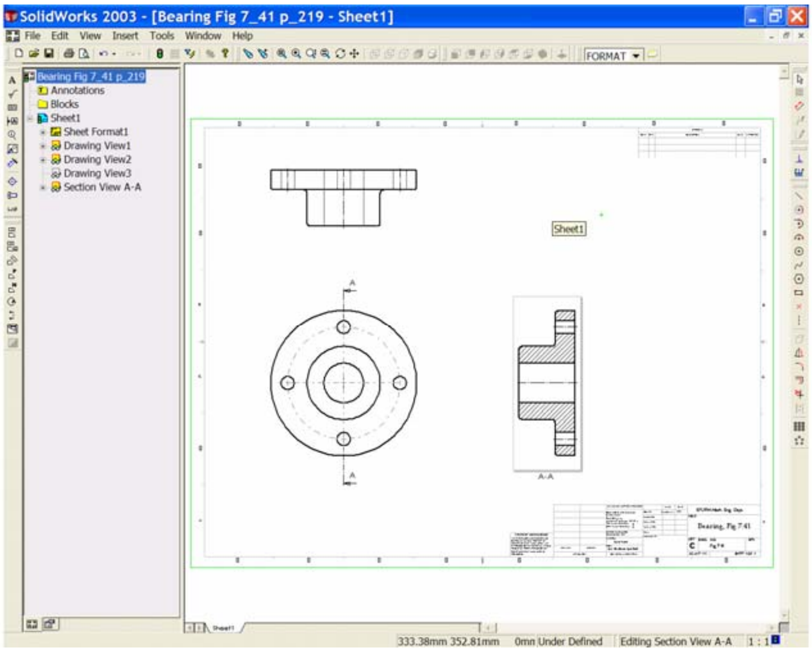


Figure 9

14. The Cutting Plane Line Font may need to be changed from the default. To do this select: **Tools**, **Options**, **Document Properties** and **Line Font**. Select '**Section Line**' and choose '**Chain**' from the '**Style**' list and set '**Thin**' for the '**Thickness**'.

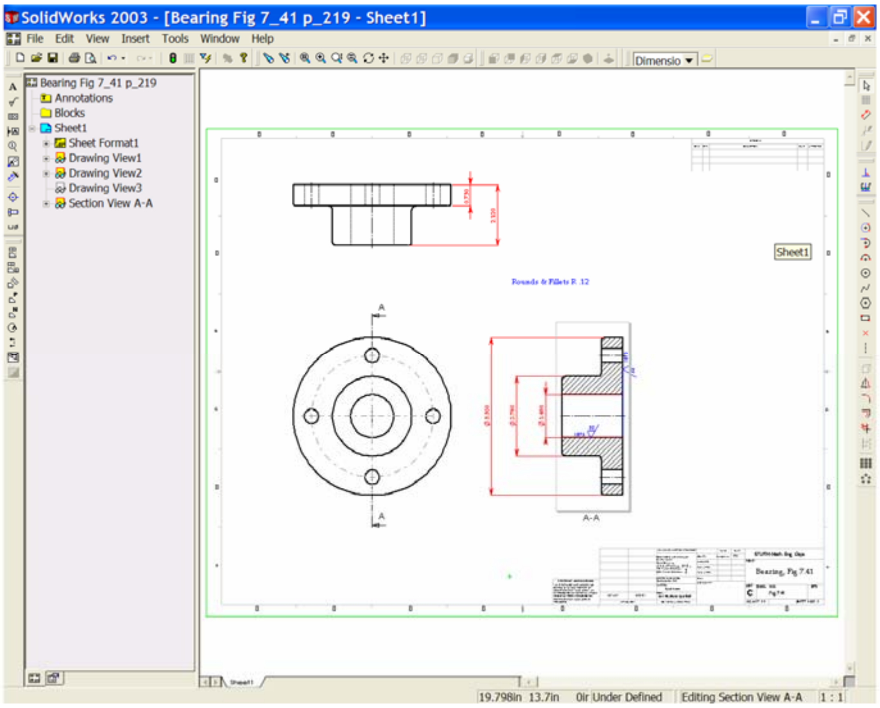


Figure 10

15. Create layers for Dimensions and for Machine Symbols. Set the machine symbols layer current and add the symbols as shown in Figure 10. The R_a value should be 32µ in. and Machine Allowance is to be 3/16" (.1875"). Use the 'Font' to control the size of the symbol.

16. We may add the data for Rounds & Fillets. Use the ‘Note’ button and add ‘Rounds & Fillets R .12’.

17. From ‘Insert’ Model Items, insert the dimensions on the dimensions layer. Re-arrange the dimensions as shown in the Figure 10.

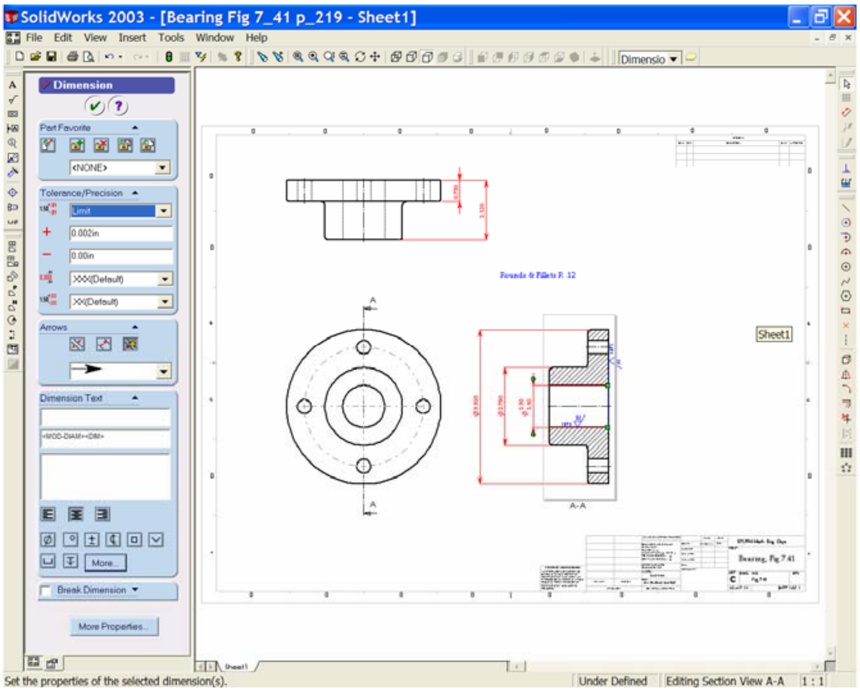


Figure 11

18. The bore dimension has a maximum and minimum limit. This may be added as follows; Select the bore diameter dimension. The Dimension Property Manager’ opens and we may select ‘**Limit**’ from the ‘**Tolerance Type**’ window.
*The base sketch used the ‘**Minimum Limit**’ diameter for the model construction, therefore we do not need to change the ‘–ve’ value. The ‘**Maximum Limit**’ for the bore is Ø1.500 therefore we need to enter 0.002 in the ‘+ve’ field. Ensure that 3 decimal places are shown in the precision fields. See Figure 11.*

    Select OK to update the dimension. Save your work!!!

19. We need to dimension the four bolt holes. Select the ‘**Dimension**’ tool , choose the hole at the top of the Elevation and place the dimension. In the Property Manager edit the dimension to remove the brackets and add the data:

    Ø0.500 Four holes
    Equally spaced on Ø4.24

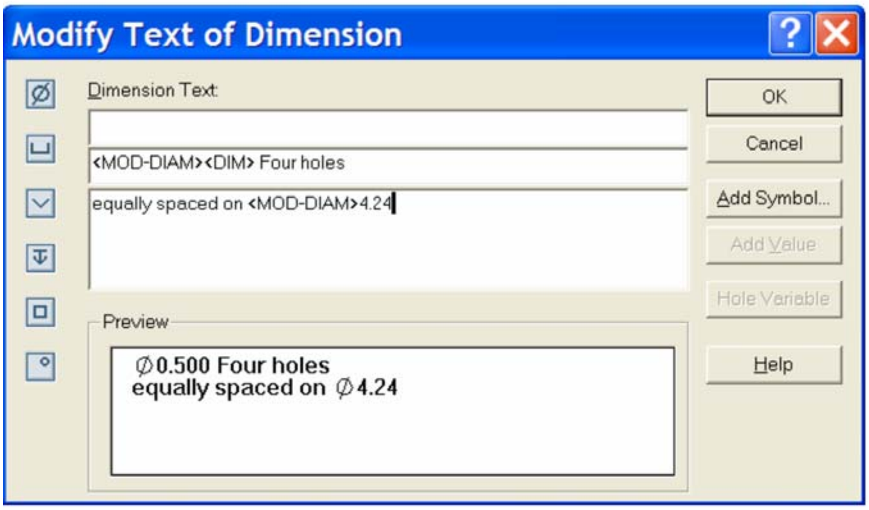


Figure 12

20. To complete the drawing add a pictorial view of the part showing tangent edges 'with font'. See figure 13.

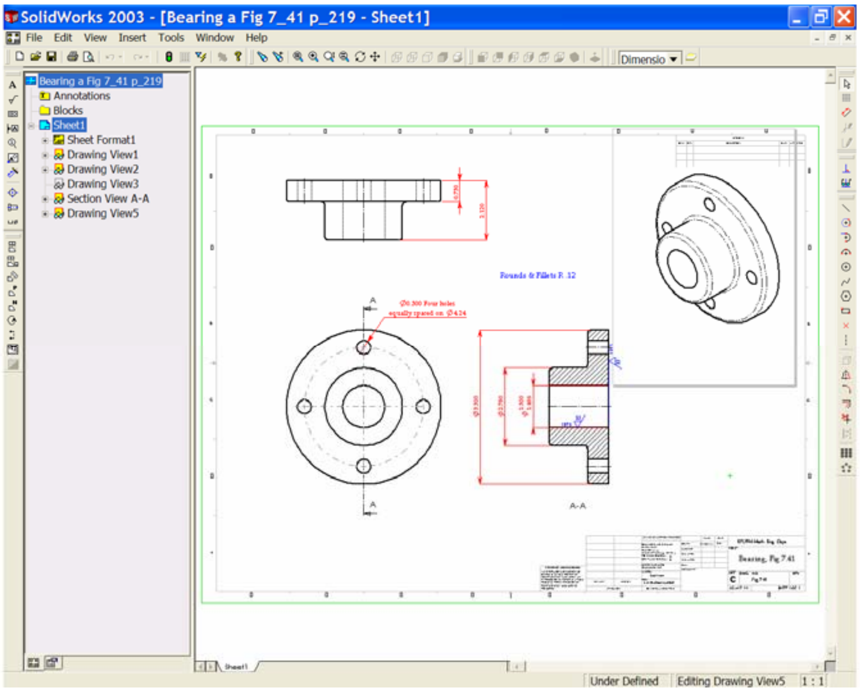


Figure 13

Module 11: Reference Planes; Vertical Bearing Support

In this Tutorial, the creation and usage of reference geometries are demonstrated. The model of Vertical Bearing Support is as shown in Figure 1.

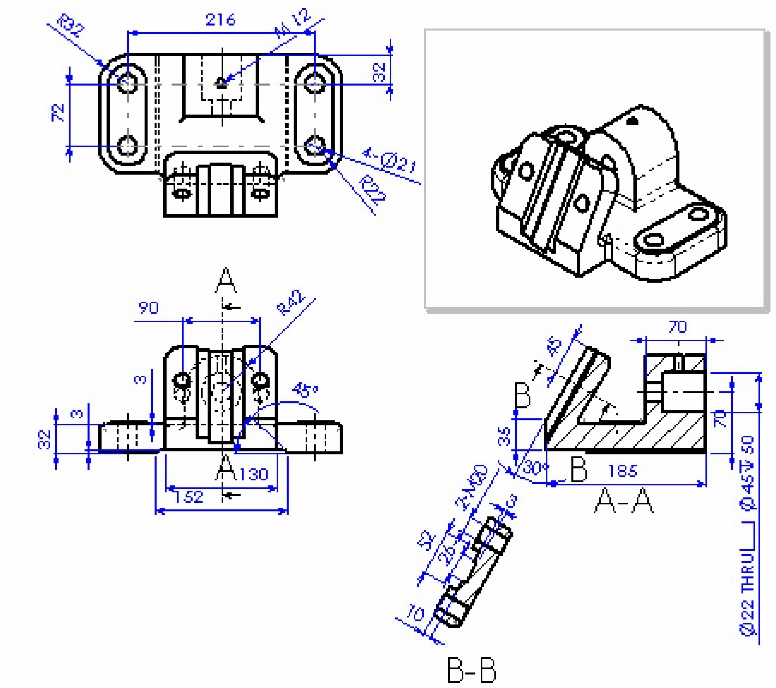


Figure 1 Vertical Bearing Support

It can be built by following the procedures as shown in Figure 2.

1. Create a base using Extruded Boss/Base.
2. Add a slot boss for creating the slot body.
3. Create a plane for slot body with **At Angle**, and then a Slot body.
4. Cut the Slots on the slot body and the base.
5. Add hole bosses for drilling holes.
6. Drill holes and add fillets.

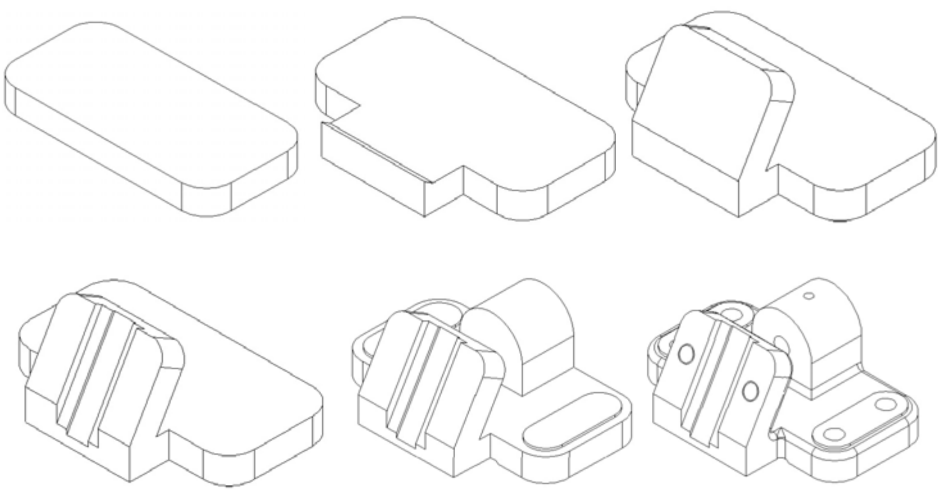


Figure 2 Modeling process of Vertical Bearing Support.

Create Support Base

1. Start Solid Works
2. Select **New** on the Standard toolbar, or click **File, New** on the menu bar.
3. Select **Part** from the **Template** tab in the dialog box, and click **OK**.
4. Click **File, Properties** on the menu bar.
5. Fill in the necessary properties in the dialog box, and click **OK** to accept the properties and close the dialog box.
6. Click **Tools, Options** on the menu bar to open the option dialog box.
7. Select the **Document Properties** tab, and click **Unit** in the properties tree text box.
8. Select **Millimeter** and set the decimal places to **2**.
9. Click **OK** to close the Properties dialog box.
10. Select **Top** Plane in **FeatureManager** design Tree and click **Normal To** to bring Top plane parallel to the screen.
11. Select **Sketch** on the Sketch toolbar to open a new sketch on the **Top** Plane.
12. Select **Rectangle** on the Sketch Tools toolbar, and draw a rectangle.
13. Select **Fillet** on the Sketch Tools toolbar to fillet the four corners of the rectangle.
14. Select **Dimension** on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 3. To change the fillet radius to **32 mm,** double click on fillet dimension, **Modify** dimension window will open, type **32** and press Enter.
15. Click **Zoon to Fit** to see the entire sketch.

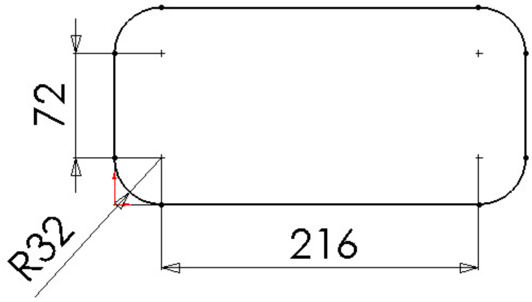


Figure 3 Base sketch and dimensions

16. Select **Extrude Boss/Base** on the Features toolbar. The Base-Extrude PropertyManager appears.
17. Select **Blind** in the End Condition drop down list box.
18. Type **32mm** in the distance 1 box as shown in Figure 4.
19. Click **OK** to finish the base extrusion.

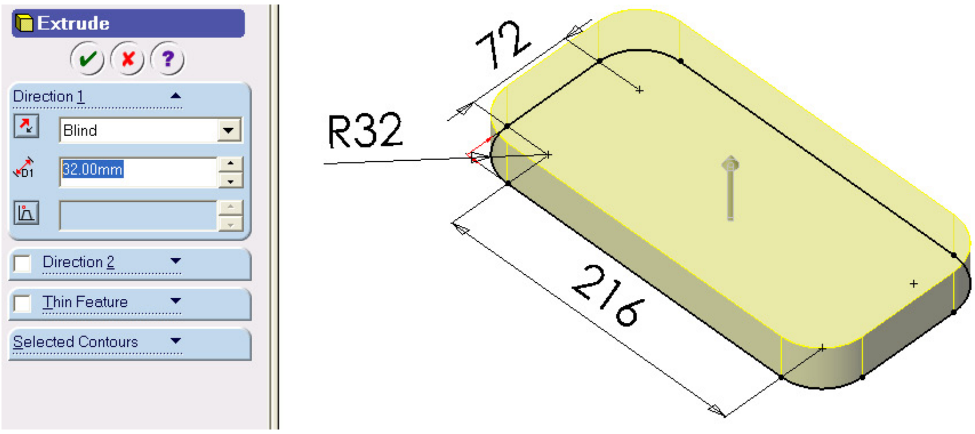


Figure 4 Base extrude

Create an Offset Plane and a Support Boss

1. Click **Plane** on the Reference Geometry toolbar, or click **Insert**, **Reference Geometry**, **Plane** on the menu bar.
2. Select **Offset Distance** on the PropertyManager.
3. Select the side face of the base as shown in Figure 5. Its name appears in the **Selection** text box.
4. Enter **140mm** in the Distance text box.
5. Select the **Reverse Direction** check box.
6. Click **OK**. The **Planel** is created and displayed in the FeatureManager Design Tree.

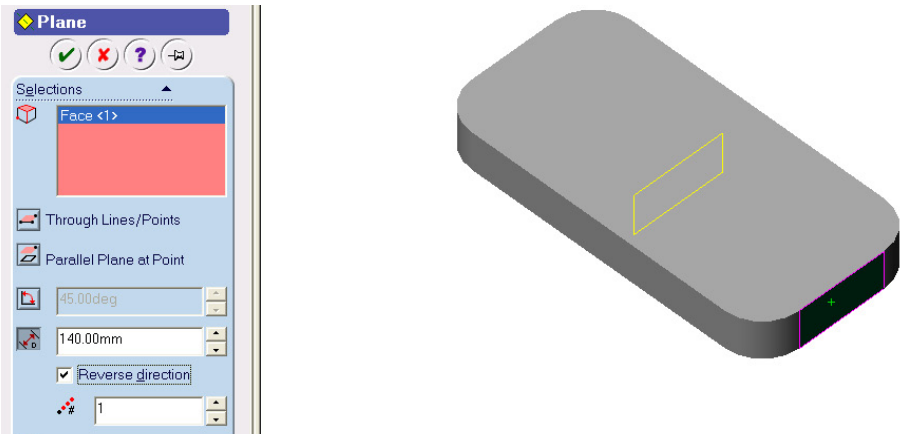


Figure 5 Offset plane

7. Select the **Planel** in the FeatureManager Design Tree or graphics window.
8. Select **Sketch** on the Sketch toolbar to open a new sketch.
9. Click **Normal To** on the Standard Views toolbar to make the sketch plane parallel to the screen.
10. Select **Line** on the Sketch Tools toolbar to draw a polygon at the left edge of the part. The right vertical line of the sketch should align with the left edge of the part. Its two endpoints should be coincident with the endpoints of left edge of the part.

11. Select **Dimension** on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 6.

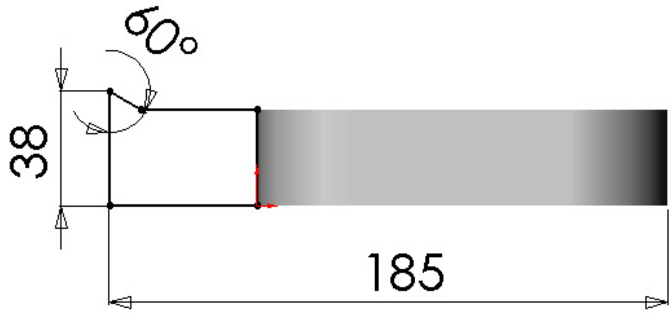


Figure 6 Boss sketch

12. Select **Extrude Boss/Base** on the Features toolbar.

13. Click **Isometric** to have 3-D visualization.

14. Select **Mid Plane** in the end condition drop down list box.

15. Type **130 mm** in the distance 1 text box as shown in Figure 7.

16. Click **OK** to finish the boss extrusion.

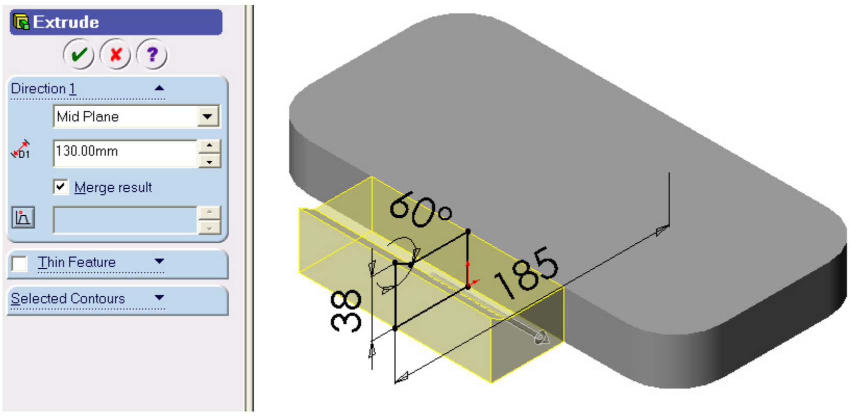


Figure 7 Boss extrusion parameters

Create an Angle Plane and a Slot Body

1. Click **Plane** on the Reference Geometry toolbar, or click **Insert, Reference Geometry, Plane** on the menu bar.

2. Click **At Angle** in the dialog box, and then select the front face of the boss in the graphics window. Its name appears in the Selections text box.

3. Select the edge of the boss tip as shown in Figure 8.

4. Type **30** in the Angle text box.

5. Click **OK**. The **Plane2** is created and displayed in the FeatureManager Design Tree.

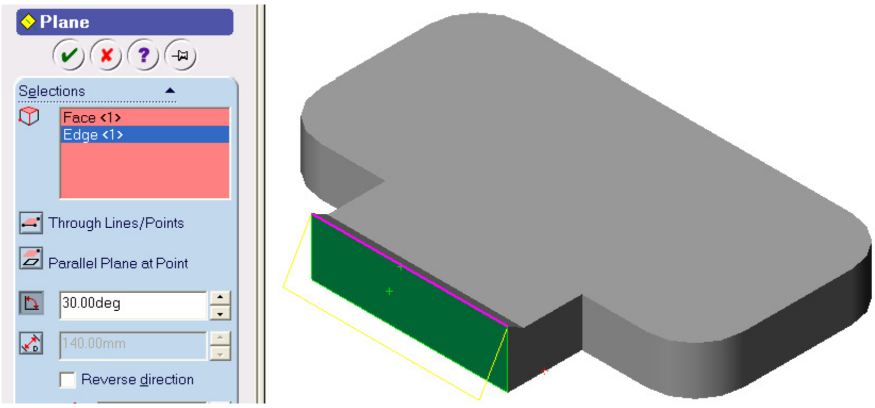


Figure 8 Plane At Angle for slot body

6. Select the **Plane2** in the FeatureManager Design Tree or in the graphics window.
7. Select **Sketch** on the Sketch toolbar to open a new sketch.
8. Click **Normal To** on the Standard Views toolbar to make the sketch plane parallel to the screen.
9. Select **Rectangle** on the Sketch Tools toolbar and draw a rectangle whose top edge aligns with the top edge of the part.
10. Select **Dimension** on the Sketch Relation toolbar. Dimension the rectangle as shown in Figure 9. If the sketch is not fully constrained, apply collinear constraint to the vertical lines and edges.

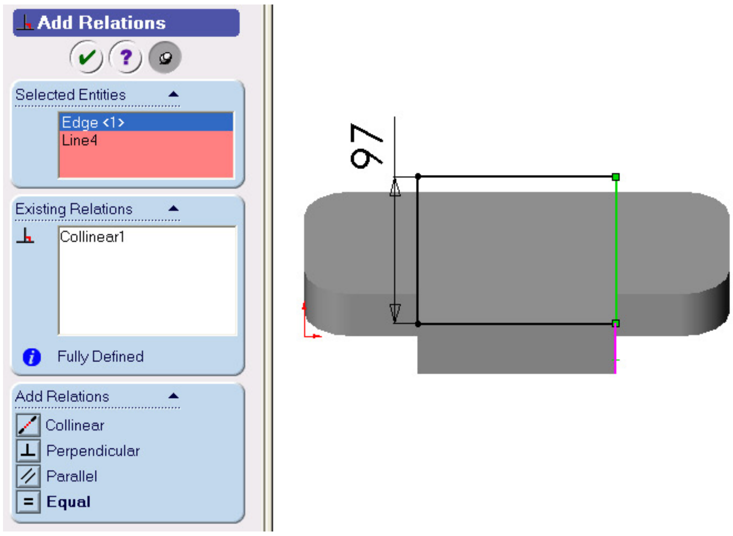


Figure 9 Slot body sketch

11. Select **Extrude Boss/Base** on the Features toolbar.
12. Select **Blind** in the end condition drop down list box.
13. Type **32 mm** in the distance 1 text box as shown in Figure 10.
14. Click Reverse Directionto flip the extrusion direction
15. Click **OK** to finish the slot body extrusion.

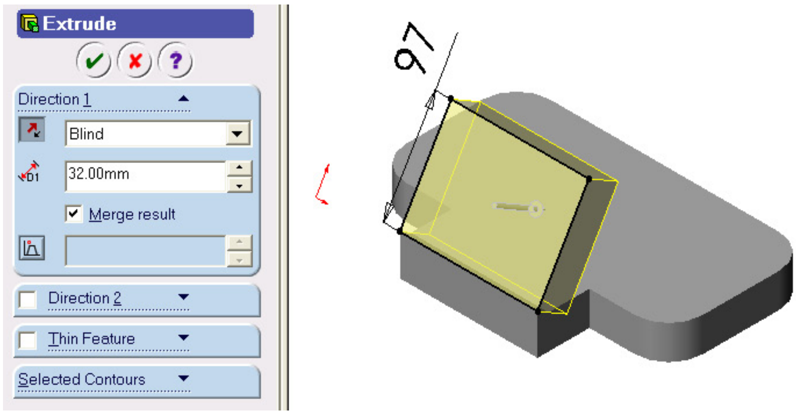


Figure 10Slot Body extrusion parameters

16. Select **Fillet** on the Features toolbar.
17. Select two top side edges of the slot body as shown in Figure 11.
18. Type **20 mm** in the Radius text box.
19. Click **OK** to finish the fillet.

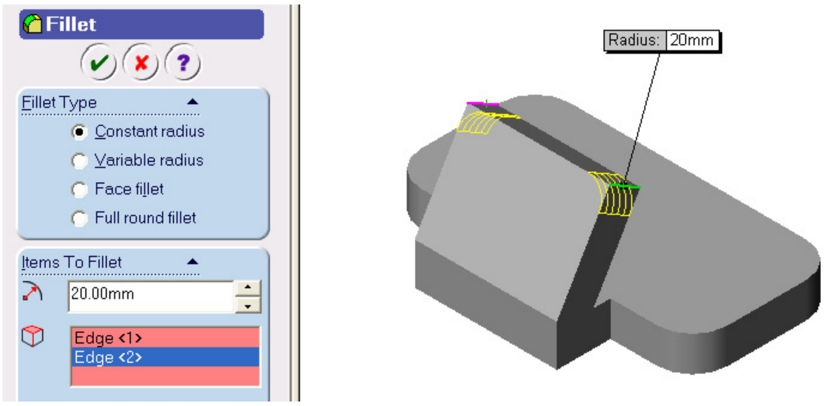


Figure 11 Slot body fillets

Cut Slots

1. Select the top face of the slot body as shown in Figure 12.

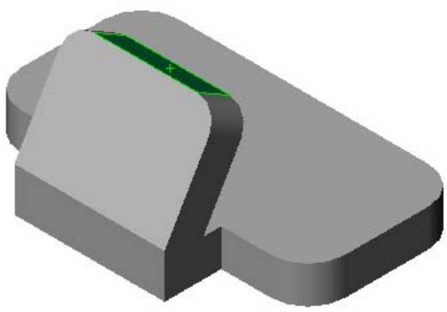


Figure 12 Sketch surface for slot cut

2. Select **Sketch** on the Sketch toolbar to open a new sketch.
3. Click **Normal To** on the Standard Views toolbar to make the sketch plane parallel to the screen.

4. Select **Centerline** on the Sketch Tools toolbar to draw a horizontal centerline as the symmetric axis of the sketch.

5. Select **Line** on the Sketch Tools toolbar to draw the sketch of the slot that is symmetric to the sketched centerline.

6. Select **Add Relation** on the Sketch Relation toolbar to add **symmetric** constraints to the sketch. (**Note:** Apply the symmetric constraint first to the centerline with the part edges to make the centerline symmetric to the part).

7. Select **Dimension** on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 13. The sketch will be fully constrained. If not, apply more constraints.

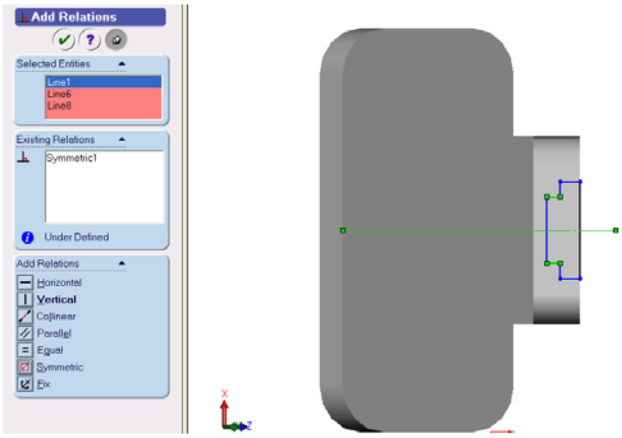


Figure 13 Slot sketch

8. Select **Extruded Cut** on the Features toolbar.

9. Select **Through All** in the end condition drop down list box as shown in the Figure 14.

10. Click **OK** to finish the slot cut.

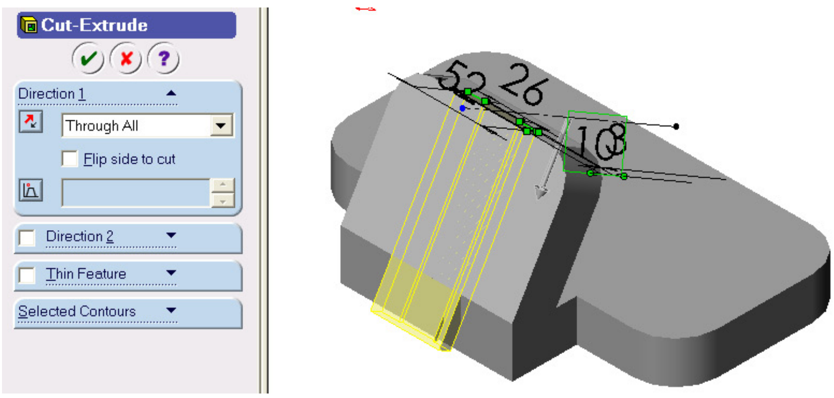


Figure 14 Slot cut parameters

11. Select back surface of the base feature as the sketch plane of the bottom as shown in Figure 15. You may rotate the part using **Rotate View** to make back surface directly visible.

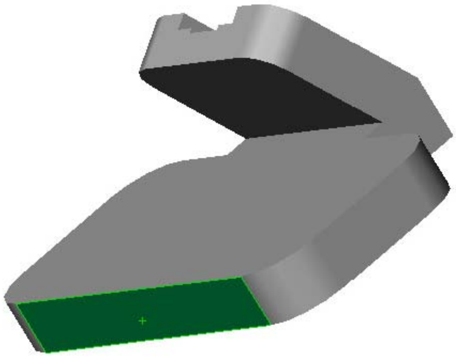


Figure 15 Sketch plane for bottom slot

12. Select **Sketch** on the Sketch toolbar to open a new sketch.
13. Click **Normal To** on the Standard Views toolbar to make the sketch plane parallel to the screen.
14. Select **Centerline** on the Sketch Tools toolbar to draw a horizontal centerline as the symmetric axis of the sketch.
15. Select **Line** on the Sketch Tools toolbar to draw a sketch of the slot that is symmetric to the sketched centerline.
16. Select **Add Relation** on the Sketch Relation toolbar to add symmetric constraints to the sketch. (**Note:** Apply the symmetric constraint first to the centerline of the part edge to make the centerline symmetric to the part).
17. Select **Dimension** on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 16. The sketch will be fully constrained. If not, apply more constraints.

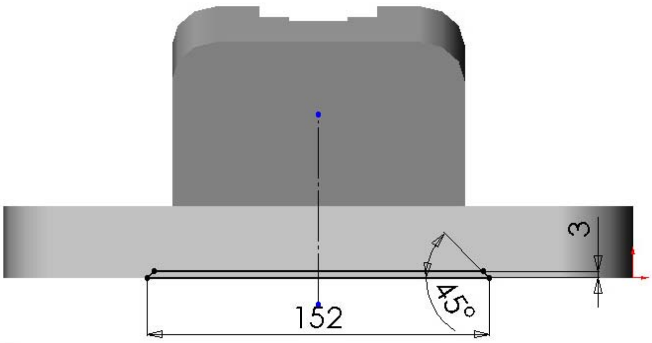


Figure 16 Bottom slot sketch

18. Select **Extruded Cut** on the Features toolbar.
19. Select **Through All** in the end condition drop down list box as shown in the Figure 17.
20. Click **OK** to finish the bottom slot cut.

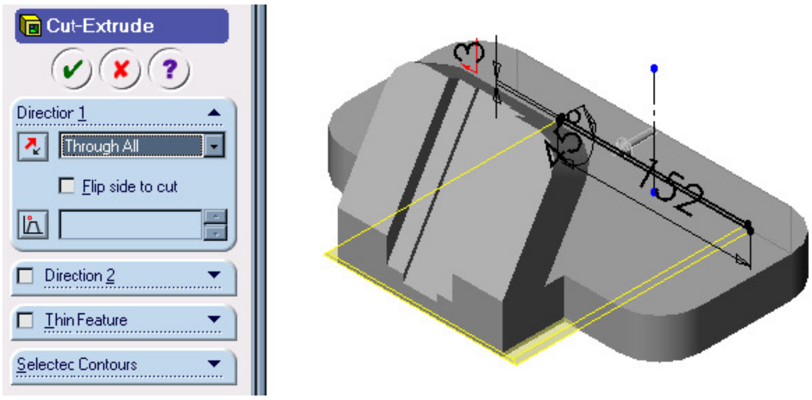


Figure 17 Bottom slot cut parameters

Create Bosses on the Base

1. Select the top face of the base as the sketch plane of the hole bosses as shown in Figure 18.

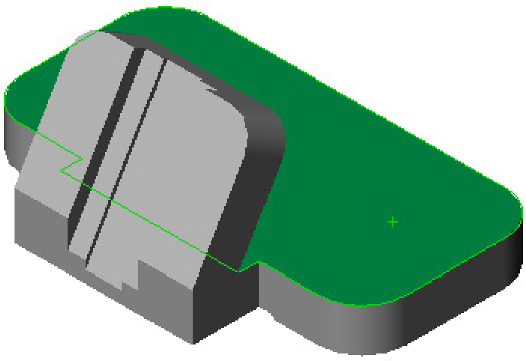


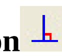
Figure 18 Sketch plane for hole boss

2. Select **Sketch**  on the Sketch toolbar to open a new sketch.

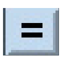
3. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.

4. Select **Centerline** on the Sketch Tools toolbar to draw a horizontal centerline as the symmetric axis of the sketch.

5. Select **Centerpoint Arc**  on the Sketch Tools toolbar, and draw two arcs (two half-circles) to form the top and bottom edges of the sketch.

6. Select **Add Relation**  on the Sketch Relation toolbar.

7. Click **Keep Visible**  on the PropertyManager to activate the command.

8. Select two arcs, and apply **equal**  constraint.

9. Select the top arc and the top edge arc of the part, and apply **Concentric**  constraint.

10. Select the bottom arc and the bottom edge arc of the part, and apply **Concentric** constraint.

11. Select the centerline and the two outside edges of the part, and apply **symmetric** constraint.

12. Select **Line** on the Sketch Tools toolbar, and draw two lines to close the sketch.

13. Select the sketch and the centerline by drawing a window around them (you may add entities to selection by holding the 'Control' key and clicking on he required entities one by one), and click **Mirror** on the Sketch Tools toolbar to mirror the sketch to the other side.

14. Select **Dimension** on the Sketch Relation toolbar. Dimension the sketch as shown in Figure 19.

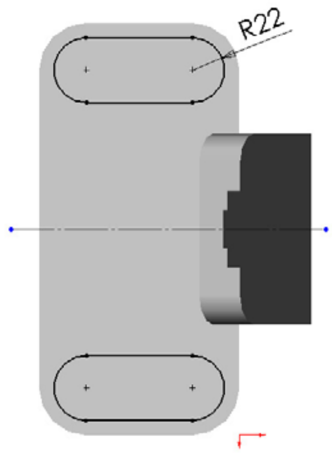


Figure 19 Hole boss sketch

15. Select **Extruded Boss/Base** on the Features toolbar.

16. Click **Isometric** to have 3-D visualization.

17. Select **Blind** in the end condition drop down list box.

18. Type **3 mm** in the **distance 1** box as shown in Figure 20.

19. Click **OK** .

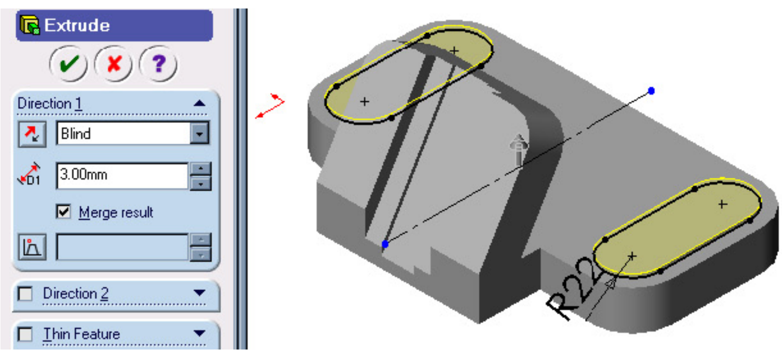


Figure 20 Hole Boss Extrusion Parameters

20. Select the back surface of the base as shown in Figure 21.

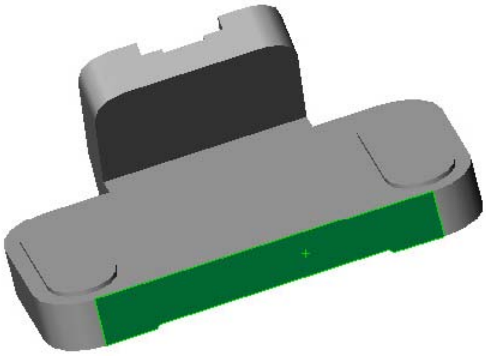


Figure 21 Sketch plane for the counterbored hole boss

21. Select **Sketch** on the Sketch toolbar to open a new sketch.
22. Click **Normal To** on the Standard Views toolbar to make the sketch plane parallel to the screen.
23. Select **Centerline** on the Sketch Tools toolbar to draw a horizontal centerline as the symmetric axis of the sketch.
24. Select **Add Relation** on the Sketch Relation toolbar. Select the centerline and the two side edges of the part, and apply **Symmetric** constraint.
25. Select **Centerpoint Arc** on the Sketch Tools toolbar, and draw a half circle whose center is coincident with the centerline.
26. Select **Line** on the Sketch Tools toolbar to draw three lines to close the sketch.
27. Select **Dimension** on the Sketch Relation toolbar, and dimension the sketch as shown in Figure 22.

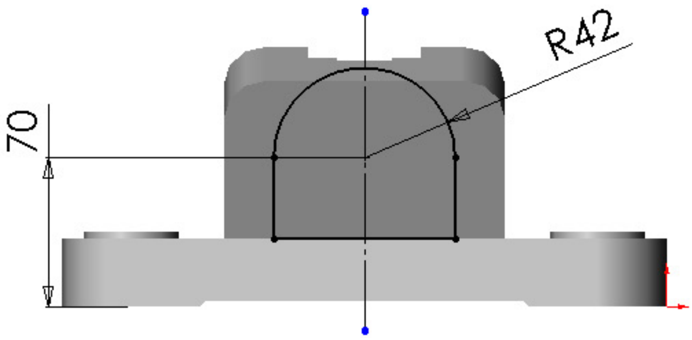


Figure 22 Sketch for counterbored hole boss

28. Select **Extruded Boss/Base** on the Features toolbar.
29. Click **Isometric** to have 3-D visualization.
30. Select **Blind** in the end condition drop down list box.
31. Type **70mm** in the **direction 1** box.

32. Click **Reverse Direction**  to flip the extrusion direction as shown in Figure 23.

33. Click **OK** .

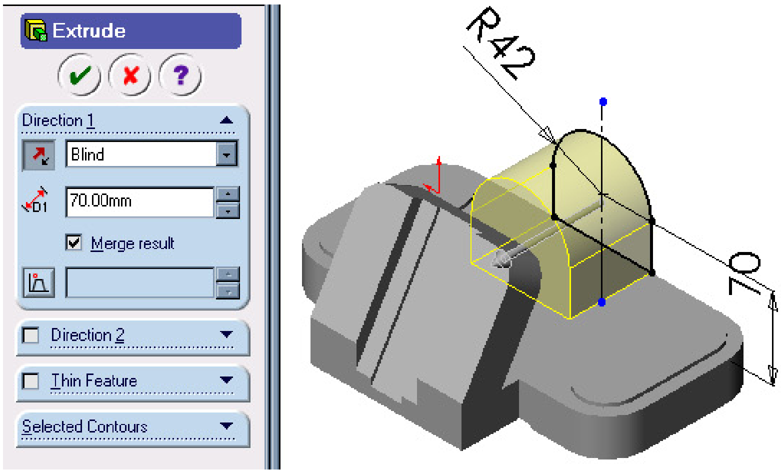


Figure 23 Counterbored hole boss

Create Holes and Fillets

1. Select the end surface of the counterbored hole boss.
2. Select the **Hole Wizard**  on the Features toolbar, and select the **Counterbore** tab.

   Note: Various types of holes can be created using corresponding tab.
3. Select **Up To Next** in the End Condition & Depth cell.
4. Type **22mm** in the Hole Fit & Diameter cell. **45 mm** and **50 mm** in the Counterbore Diameter & Depth cells as shown in Figure 24.
5. Click Next.

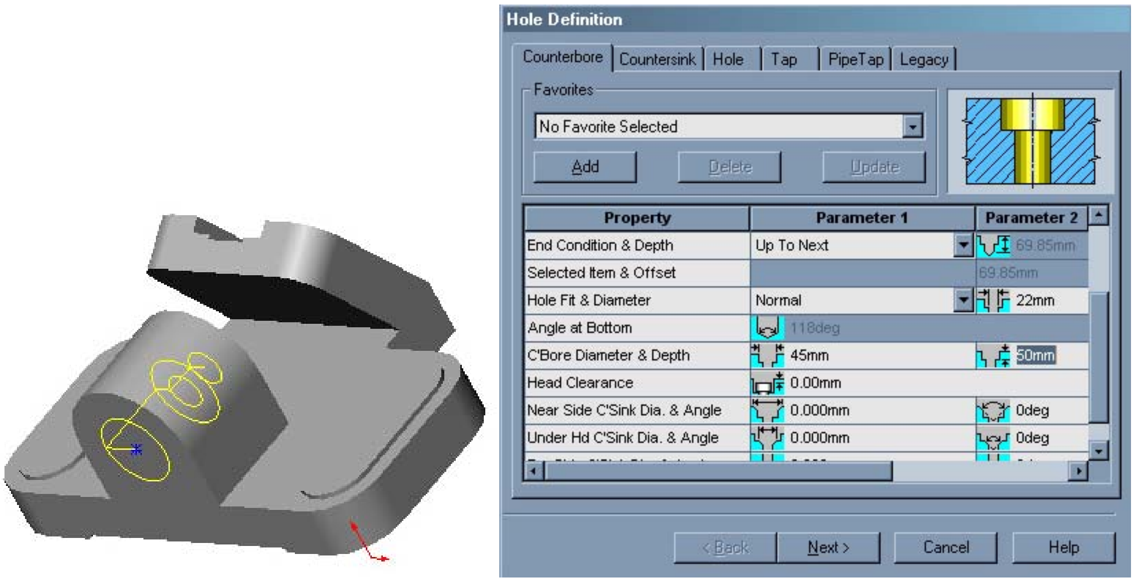


Figure 24 Counterbored Hole Parameters

6. Select **Add Relation** on the Sketch Relation toolbar.
7. Select the hole center point and the arc edge of the boss, and click **Concentric**.
8. Click **Finish** to create the counterbored hole.
9. Select the hole boss surface as shown in Figure 25.

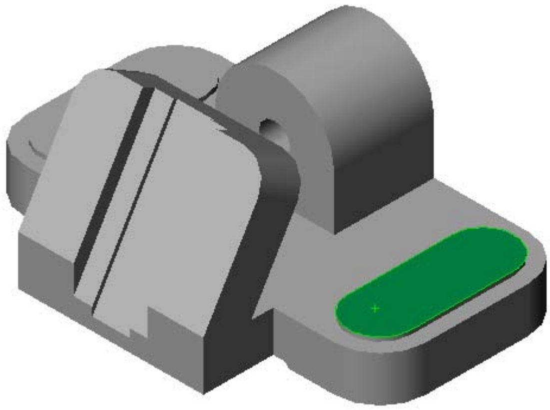


Figure 25 Sketch Plane for Base Holes

10. Select **Sketch** on the Sketch toolbar to open a new sketch.
11. Click **Normal To** on the Standard Views toolbar to make the sketch plane parallel to the screen.
12. Select **Circle** on the Sketch Tools toolbar, and draw **one** circle for the hole.
13. Select **Add Relation** on the Sketch Relation toolbar, and add concentric constraint to the circle with the arc edge of the base.

    Remark: You may similarly draw **three** other circles, Add **Equal** constraint to the circles, but we will use another useful command to produce these circles.)
14. Select **Dimension** on the Sketch Relation toolbar. Dimension the circle diameter to **21mm** as shown in Figure 26.

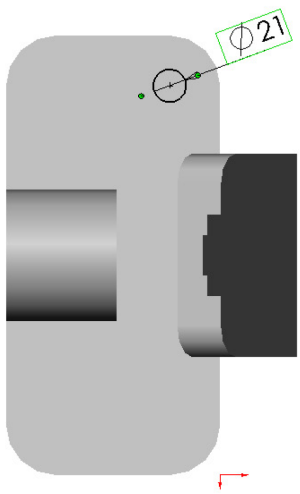


Figure 26 Base Hole Sketch

15. Select **Extruded Cut** on Features toolbar. Select **Through All** in the end condition drop down list box as shown in Figure 27.

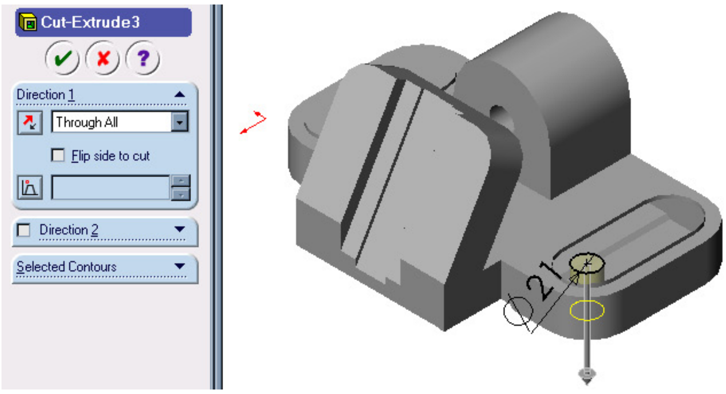


Figure 27 Base holes

16. Click **0K** .

Copying Holes using Linear Pattern

17. Make sure that Hole you have just created is selected in FeatureManager Design Tree. If not, select it by Clicking on it either in Graphic area or from FeatureManager Design Tree.

18. Click **Linear Pattern** on Feature Toolbar, Linear Pattern Prperty Manager opens as shown in Figure 28.

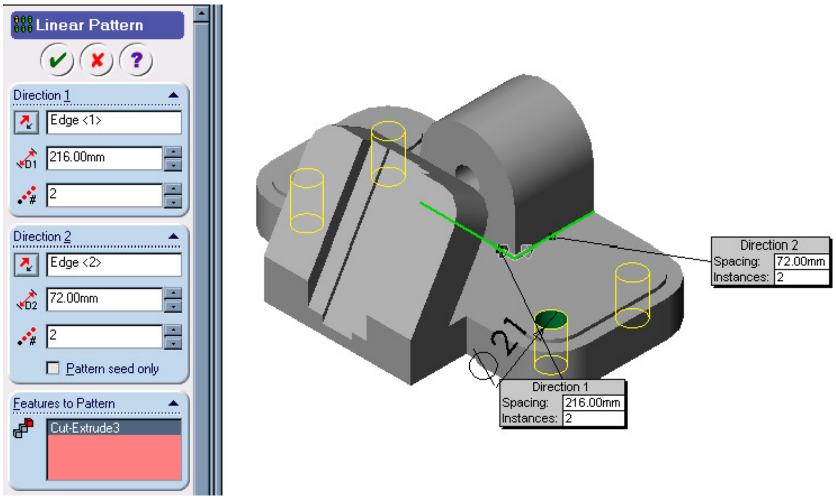


Figure 28 Copying Features using Linear Pattern.

19. Click on any edge parallel to Direction 1 as shown in Figure 28.

20. Type **216 mm** for **Spacing** .

21. Type **2** for **Number of Instances** . A preview of the copied hole will appear as shown in Figure 28. Make sure that it is being copied in correct direction. If not, click on **Reverse Direction**.

22. Click on any edge parallel to Direction 2 as shown in Figure 28.

23. Type **72 mm** for **Spacing** .

24. Type **2** for **Number of Instances** . A preview of the copied hole will appear as shown in Figure 28. Make sure that it is being copied in correct direction. If not, click on **Reverse Direction** .

25. Click OK .

26. Select the end face of the counterbored hole boss as shown in Figure 29. You may need to rotate the part.

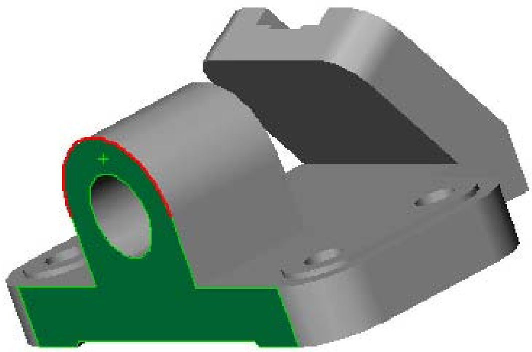


Figure 29 Sketch Plane for Reference Point

27. Select **Sketch** on the Sketch toolbar to open a new sketch.

28. Click **Normal To** on the Standard Views toolbar to make the sketch plane parallel to the screen.

29. Select **Point** on the Sketch Tools toolbar, and draw a point on the top edge of the counterbored hole boss.

30. Select **View, Temporary Axes** on the menu bar to show the temporary axis for adding relation.

31. Select **Add Relation** on the Sketch Relation toolbar.

32. Select the point and the edge. Click **Coincident** to make the point coincident with the arc edge.

33. Select the hole axis and the point as shown in Figure 30. Select **Vertical** **.**

34. Click **Rebuild** on the Standard toolbar to exit sketch.

35. Click **Plane** on the Reference Geometry toolbar, or click **Insert, Reference Geometry**, **Plane** on the menu bar.

36. Select **On Surface** **.**

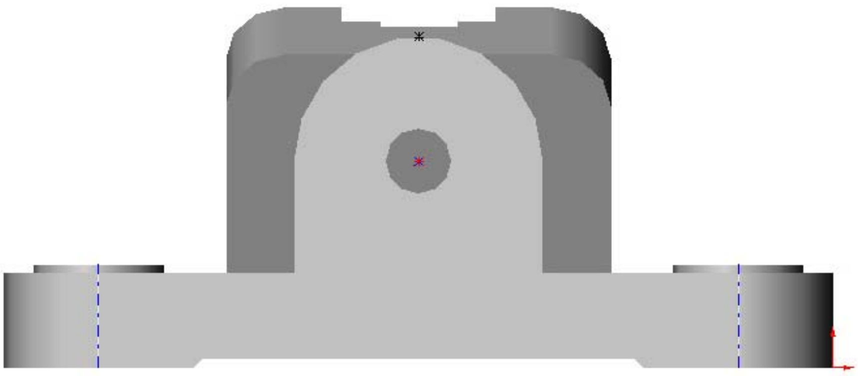


Figure 30 Sketch Point for Creating On Surface Plane

37. Select the cylindrical face of the boss and the sketch point as shown in Figure 31. Their names appear in the Selections text box.

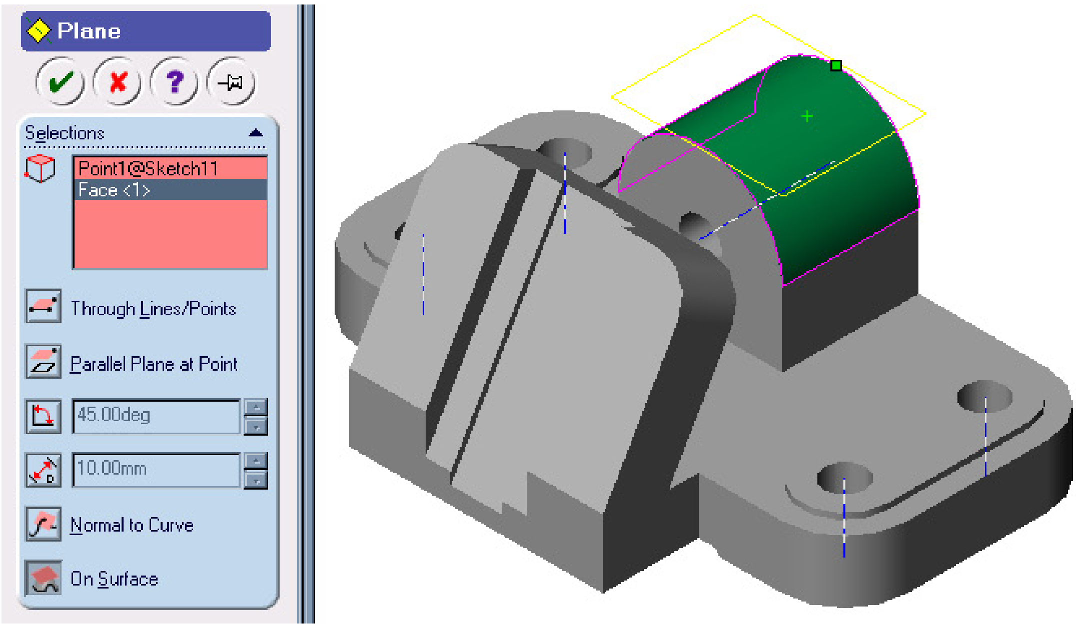


Figure 31 Top hole placement plane

38. Click **OK** **.** The **Plane3** is created and displayed in the FeatureManager Design Tree.

39. Select **Plane 3** in the Feature Manager Design Tree or in the graphics windows.

40. Select **Hole Wizard**  on the Features toolbar.

41. Select **Tap** tab in the Hole Definition dialog box.

42. Select the parameters as shown in Figure 32. Click **Next**.

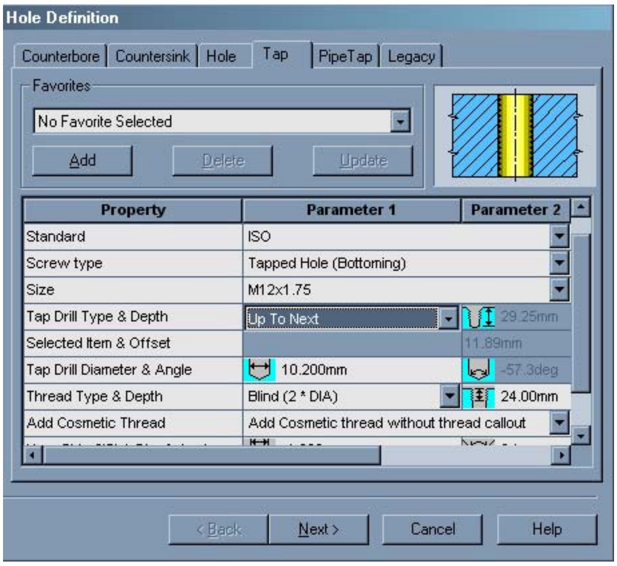


Figure 32 Top hole definition.

43. Click **Normal To**  on the Standard Views toolbar to make the sketch plane parallel to the screen.

44. Add constraints and dimension the hole center as sown in Figure 33. Then click **Finish** in the Hole Placement dialog box. (**Note:** There will be an error message in the dialoge box. Click **OK** to ignore it.)

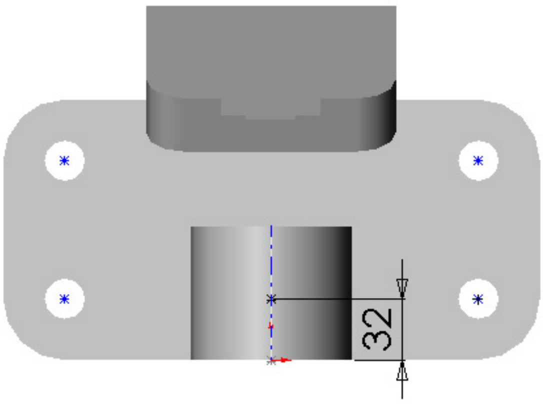


Figure 33 Top hole placement.

45. Select the front face of the slot body as shown in Figure 34.

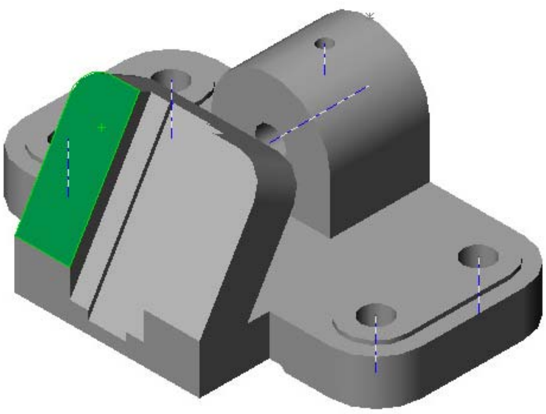


Figure 34 Slot hole sketch plane

46. Select **Hole Wizard**  on the Features toolbar.

47. Set up the Hole Definition Dialog box as shown in Figure 35. Then click **Next**.

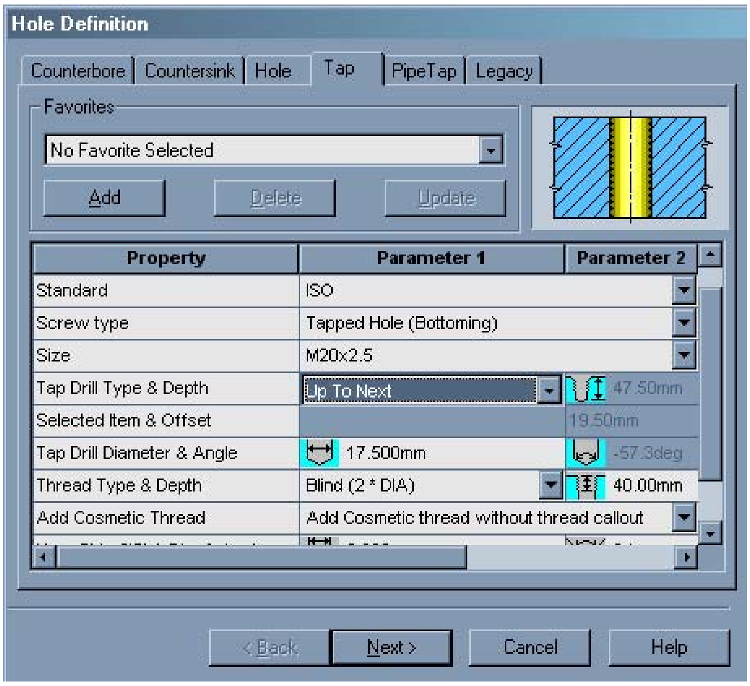
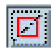


Figure 35 Slot hole definition

48. Sketch a centerline and a point for the other hole. Add Symmetric  relation between two points and centerline. Dimension the hole centers as shown in figure 36.

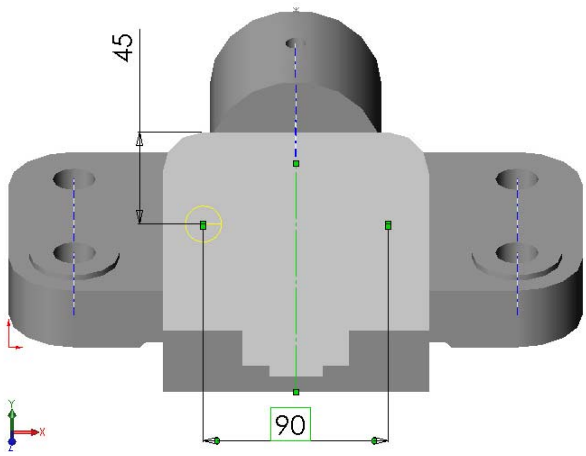


Figure 36 Slot hole placement.

49. Click Finish in the Hole Placement dialog box.

50. Add **fillets** as shown in Figure 37 and Figure 38 by using **Fillet**  on Feature toolbar. Use fillet radius as **6 mm.** Final model for Vertical Bearing Support is shown in Figure 39.

51. **Save**  the file.

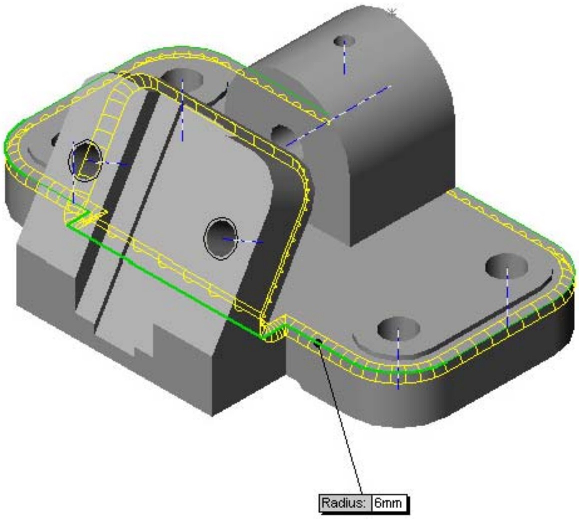


Figure 37 Base edge fillet.

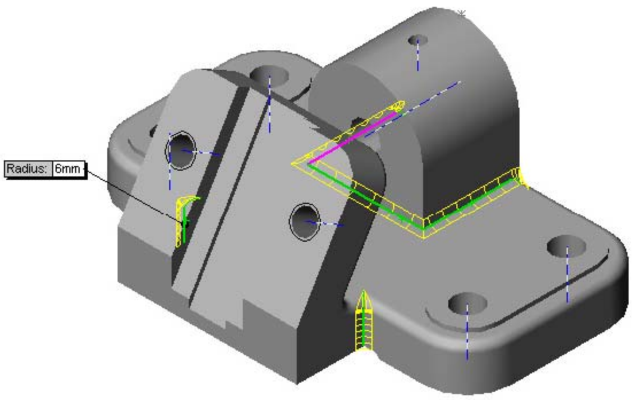


Figure 38 Slot body fillet.

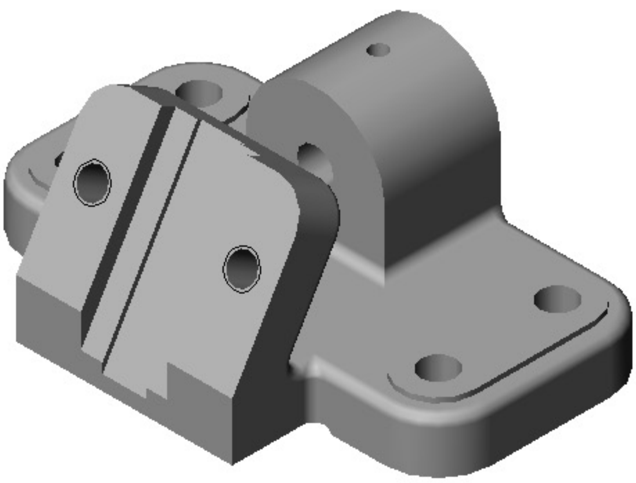


Figure 39 Vertical bearing support model

Module 13: Sweep Features II: Candle Holder

The Candle Holder is shown in Figure 1.

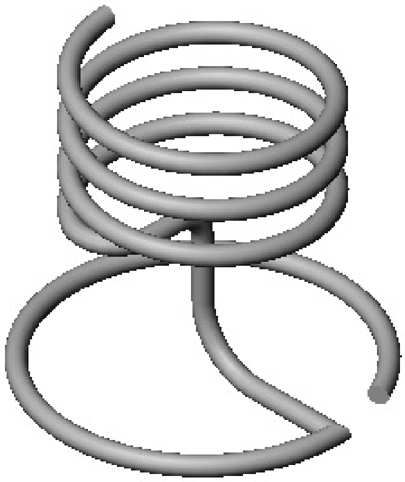


Figure 1 Candle Holder Model

It can be created by following steps:

1. Create the individual sketches that make up the main base of the candle holder. Each sketch will be a simple 2D sketch.
2. Draw a helix curve.
3. Join using the Composite Curve command, the 2D sketch geometry and a helix will form a complex 3D path for the swept feature.

To Create Model of the Candle Holder

1. Start SolidWorks.
2. Select **New** on the Standard toolbar, or click **File, New** on the menu bar.
3. Select **Part** from the **Template** tab in the dialog box, and click **OK.**
4. Click **Tools, Options** on the menu bar to open the option dialog box.
5. Select the **Document Properties** tab.
6. Click **Unit** in the properties tree text box.
7. Select **inches** and set the decimal places to 3.
8. Click **OK** to close the Properties dialog box.
9. Select **Top plane** in the PropertyManager and click **Top** on Standard View Toolbar.
10. Select **Sketch** on the Sketch toolbar to open a new sketch.
11. Create and dimension the sketch shown in Figure 1. The sketch consists of a **line** and an **arc**. Make sure to fully define the geometry.

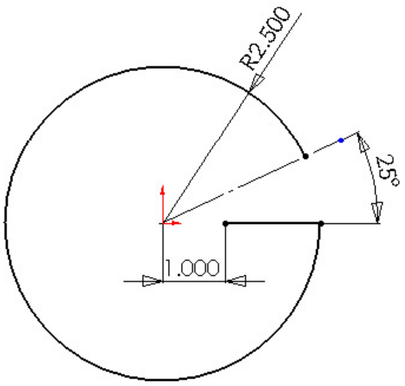


Figure 2 Creating the first sketch for the candle holder.

12. Exit the first sketch.
13. Create a new sketch on the **Front** plane. Sketch the geometry is shown in figure 3. Add **tangent** relationships to the arc to fully define it. The arc must be coincident to the end of the first sketch, which is also visible in figure 3.

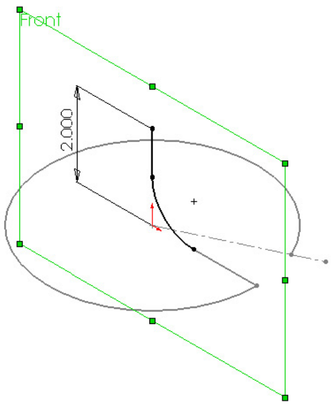


Figure 3 Creating the second sketch on the Front plane.

14. Exit the second sketch.
15. Start a new sketch on the **Righ**t plane and create a sketch shown in figure 4, which shows the completed sketch as seen from a **Right**  view. Once again, tangent relationships between the arc and adjacent geometry are critical to the design intent of the model. Make sure the start-point arc is coincident to the endpoint of the line in the second sketch.

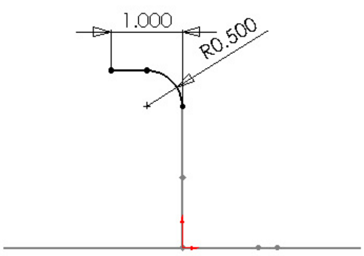


Figure 4. Developing the third sketch.

16. Exit the third sketch.

    A plane will be created for the next sketch, and the sketch geometry itself will be slightly more complex. A construction circle will be used in this next sketch, which will serve two functions. It will serve to define the location of an arc, and it will be used to define a circle for the helix. Continue with the following steps to see how this will pan out.

17. Click **Plane** on Reference Geometry Toolbar. Define a plane using the **Parallel Plane at Point** option. Select the **point** and **Top** plane for **Selections** as shown in Figure 5.

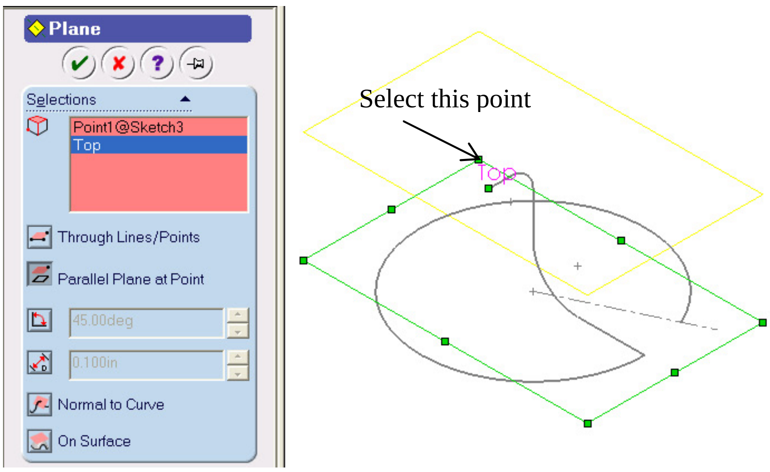


Figure 5 Reference plane parameters.

18. **Plane 1** will appear in Feature manager Design Tree. Rename the new plane as **Helix_ Plane**.

19. Select **Helix_ Plane,** click **sketch** and then click **Top** on Standard View Toolbar.

20. Draw a **circle** with center at origin. Check **For construction** box in Circle Property Manager. Circle will be converted to construction circle. Dimension it as **3.6 in**.

21. Add two arcs as shown in Figure 6. First arc is tangent to Sketch3, second arc is tangent to first arc and its radius is equal to construction circle. Add proper relations to fully define this sketch. Figure 7 shows the same sketch viewed as isometric.

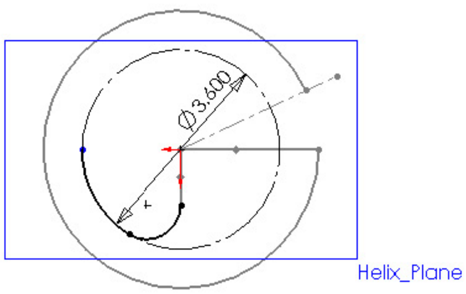


Figure 6 Developing the fourth sketch.

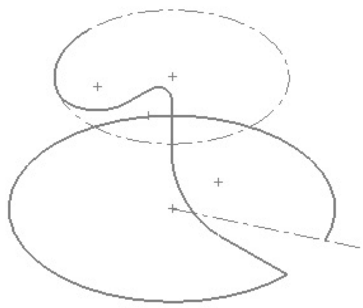
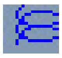


Figure 7 Fourth sketch's isometric view.

22. **Exit** the sketch.

23. Start a new **sketch**  on the **Helix_Plane** once again.

24. Select the construction circle from the fourth sketch and convert it to a sketch circle by clicking on the **Convert Entities**  sketch tool. The construction circle will be converted into a regular circle at this point.

25. Enter the **Helix**  command.

26. Use the following parameters to define the helix:

    — Pitch = .75 inch

    — Revolution = 3

    — Starting angle = 270 degrees

    — Clockwise is selected

    — Reverse Direction and Taper Helix are both unchecked

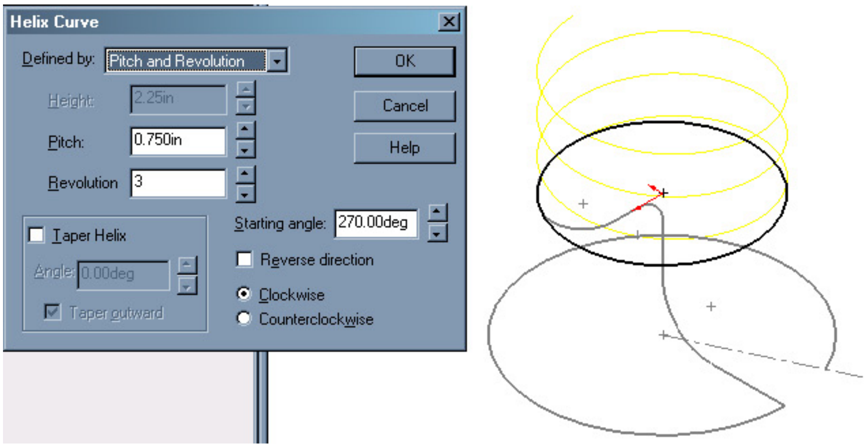


Figure 8 Helix curve parameters and helix preview.

27. Click on **OK** to create the helix once all parameters have been specified.

    You should now have something similar to that shown in figure 8. FeatureManager should list four separate sketches, along with a helix. The helix plane and other usual items will be listed also, hut they do not concern us. What needs to happen next is to join the four sketches and the helix to form a single curve.

28. Enter the **Composite Curve**  command.

29. Select the four sketches and the helix. This can be done via the work area or FeatureManager. (Hint: you can use the flyout FeatureManager by clicking on the name of the command, **Composite Curve** , at the top of PropertyManager.)

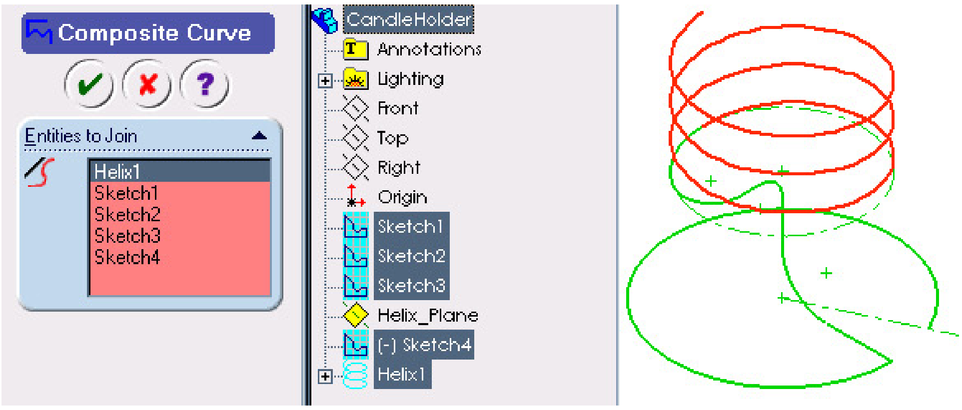


Figure 9 Composite curve parameters with FeatureManager flyout and preview of the composite curve.

30. Click on **OK**  to complete the composite curve. The result will look similar to that shown in Figure 9, but will be a single curve.

31. Click **Plane**  on the Reference Geometry toolbar.

32. Click **Normal to Curve**  in the dialog box, and then select the Composite curve and the upper end point on the composite curve in the graphics area. Preview of the plane will appear as shown in Figure 10.

33. Click **OK**  to establish the plane. **Plane1** will be listed in FeatureManager Design Tree. You may leave the same name but rename it as **Profile_Plane**.

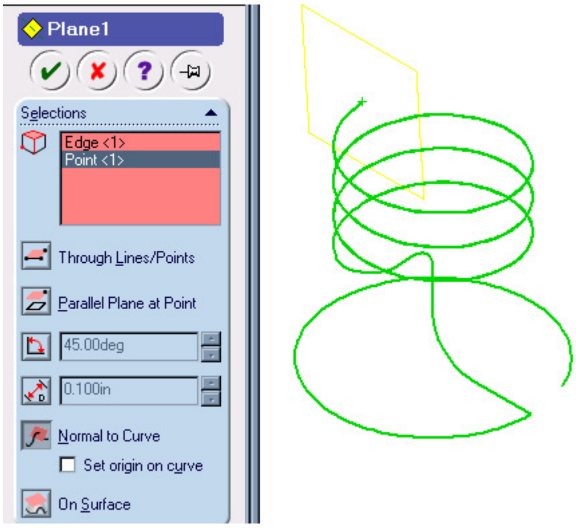
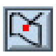


Figure 10 Profile Plane parameters.

34. Select the **Profile_Plane,** open **Sketch**  and draw **Circle**  with diameter of **.3125 in**. on it. The circle will be used as the sweep profile, and is shown in Figure 11.

35. Add a **Pierce**  constraint between the **center of the circle** and the **helix**. It is important to select the helix near the end of the helix but not the actual endpoint.

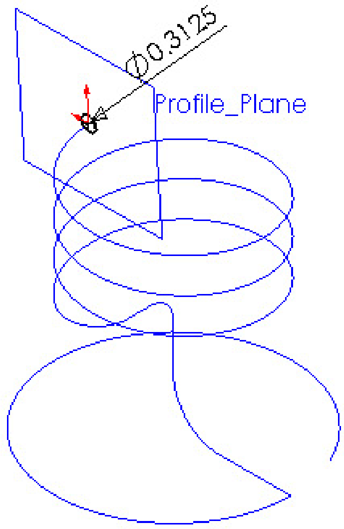


Figure 11 Profile Plane with profile sketch.

36. **Exit** the sketch.

37. Enter the **Sweep**  command.

38. Select the circle as the **Profile**  , and the composite curve as the **Path**  as shown in Figure 12.

39. Leave all default settings and click on **OK**  to complete the sweep.

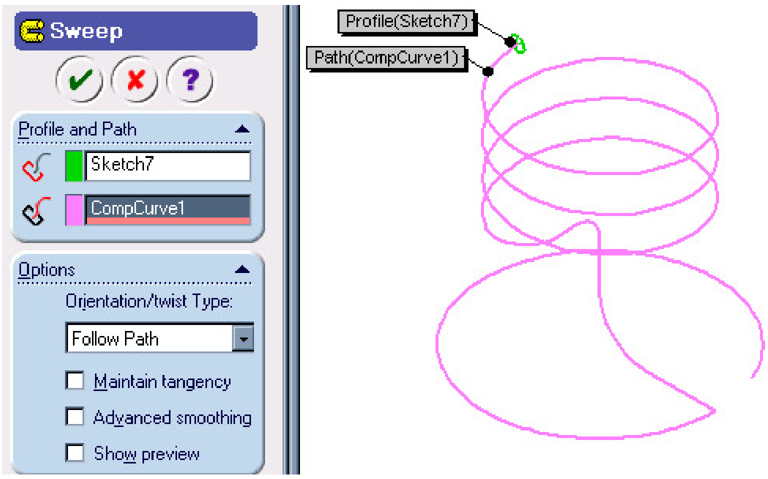


Figure 12 Sweep parameters.

40. The final feature should look similar to that shown in figure 13.

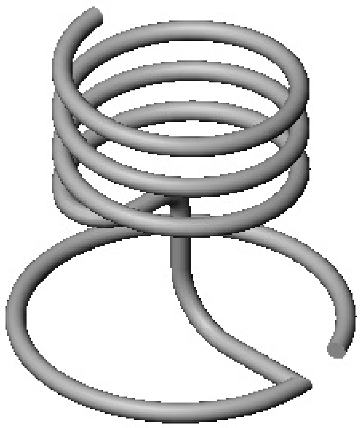


Figure 13 Candle Holder model as Sweep Feature.

Feel free to add any other finishing details, or make some modifications to the model if you like. For example, you may decide to add a sketch fillet to the first sketch to remove the sharp corner from the sweep path. Just make sure you use a radius larger than the object being swept (the .3125-inch-diameter circle). You could also add a slight outside taper to the helix, or perhaps round off the upper end of the swept feature with a revolved feature. This would help keep wax from being scraped off as a candle is rotated into the holder.

Module 14: Welded Assembly of Lofted Features

1. Open a new part file; check that the units are set to mm.

2. Open a sketch on the '**Top Plane**' and sketch a circle with a Ø100mm centred at the origin.

3. Extrude this sketch to a thickness of 12mm.

4. Select the '**Top Surface**' of the Flange and add a 3mm fillet.

5. Open a sketch on the '**Top Surface**' of the Flange and draw a vertical centre line up from the origin. See Figure 1.

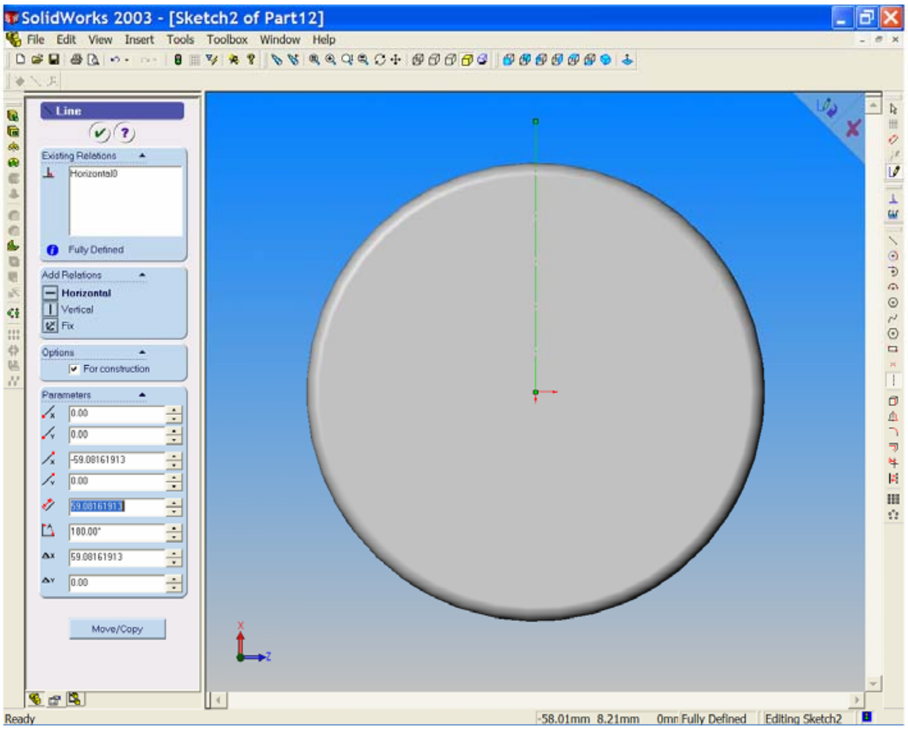


Figure 1

6. Select the '**Hole Wizard**'  from the '**Features Toolbar**'. The '**Hole Definition Window**' opens.

   - Choose the '**Countersink Tab**'.
   - For '**Standard**' choose '**ISO**'.
   - For '**Screw Type**' choose '**CTSK Flat Head**'.
   - For '**Size**' choose '**M8**'.
   - For '**End Condition & Depth**' choose '**Through All**'.
   - Leave all other values as default sizes.

   See Figure 2.

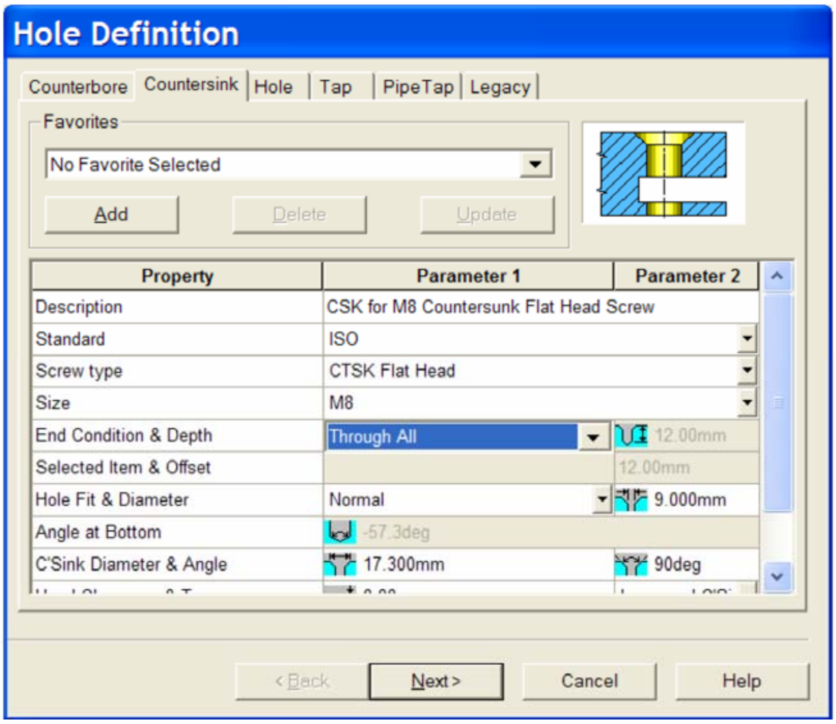


Figure 2

7. Select the '**Next**' button. The '**Hole Placement**' window opens.

Note the symbol on the cursor.

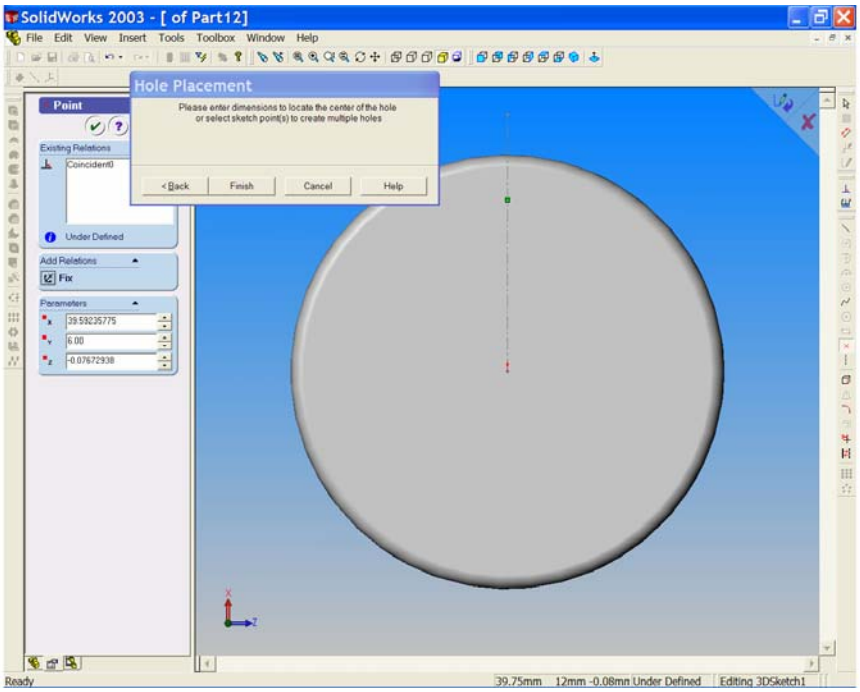


Figure 3

Place the cursor on the centre line as shown in Figure 3.

Notice that the centre line changes to ***red****.*

8. Right click and choose ‘**Select**’. Choose the dimension tool and add the distance from the origin to the hole centre. This distance should be 35mm.

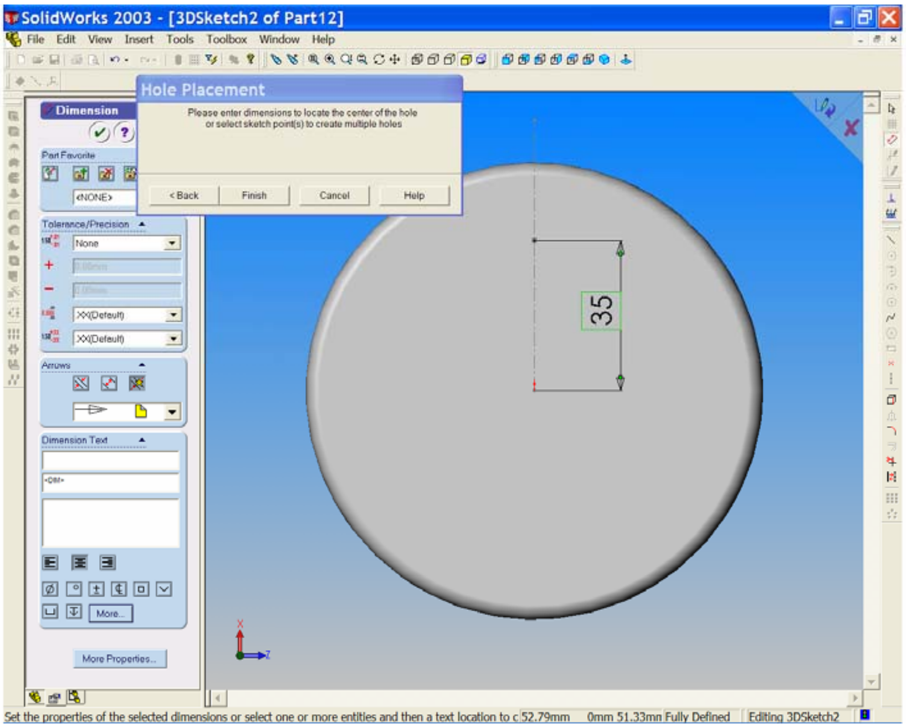


Figure 4

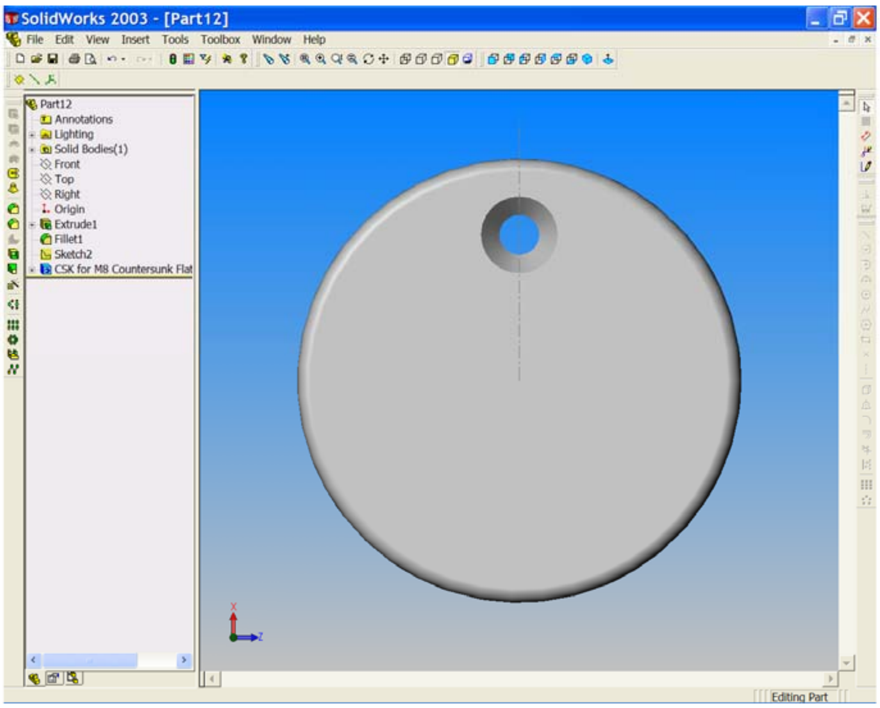


Figure 5

9. Select ‘**Finish**’ from the ‘**Hole Placement**’ window. The hole is generated and the feature is added to the design tree. See Figure 4 & 5.

10. We will now add 5 more countersunk holes creating a ‘**Circular Pattern**’ of the first hole. Choose the ‘**Circular Pattern**’ button.

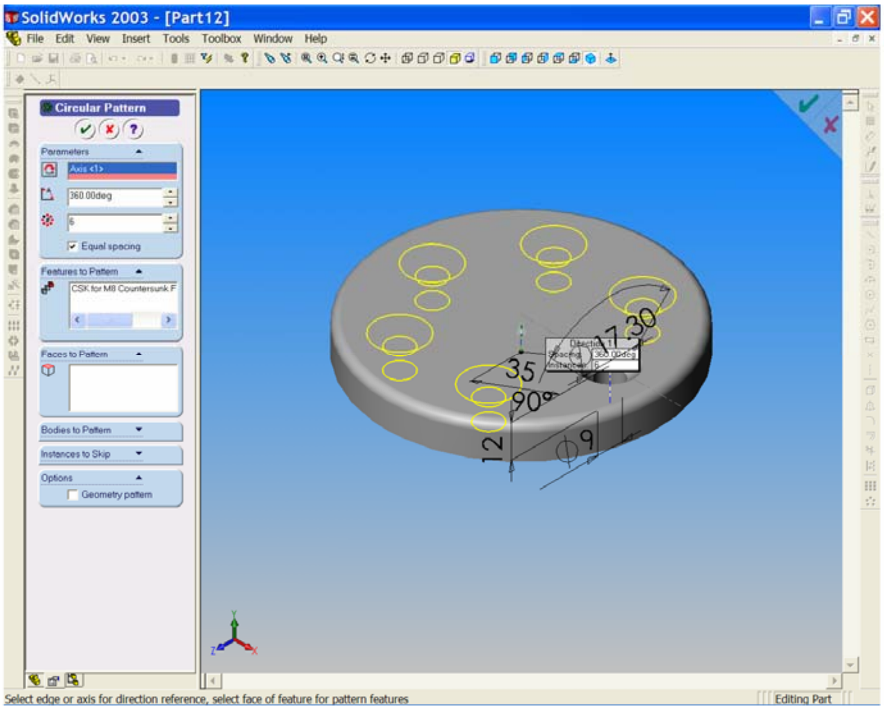


Figure 6

11. The '**Circular Pattern Property Manager**' opens.

    *Note that the first hole is already listed in the '**Features to Pattern**' field.*

    From the '**View**' drop down menu select '**Temporary Axes**'. Select '**Isometric**' to get a better view of these axes. For the '**Pattern Axis**' field (the pink field) choose the centre line in the middle of the Flange (not the centre line in the middle of the hole). Ensure that the equal spacing box is ticked and that the angle is 360°. Increase the number of instances from 2 to 6. You may use the '**up**' arrow and watch the yellow preview indicating the result as more holes are added. When all six holes are shown, select OK to complete the command. The circular pattern of countersunk holes is added to the Flange and listed in the design tree. See Figure 6.

12. Switch off the '**Temporary Axes**'. Right click on the centre line sketch and choose '**Hide Sketch**'. The Base Flange is now complete.

13. Use '**Save As**' and save the '**Flange**' for later use. You may close the file.

14. Open a new part and save as '**Lofted Column**'.

15. Activate the '**Top Plane**' and choose '**Isometric**' .

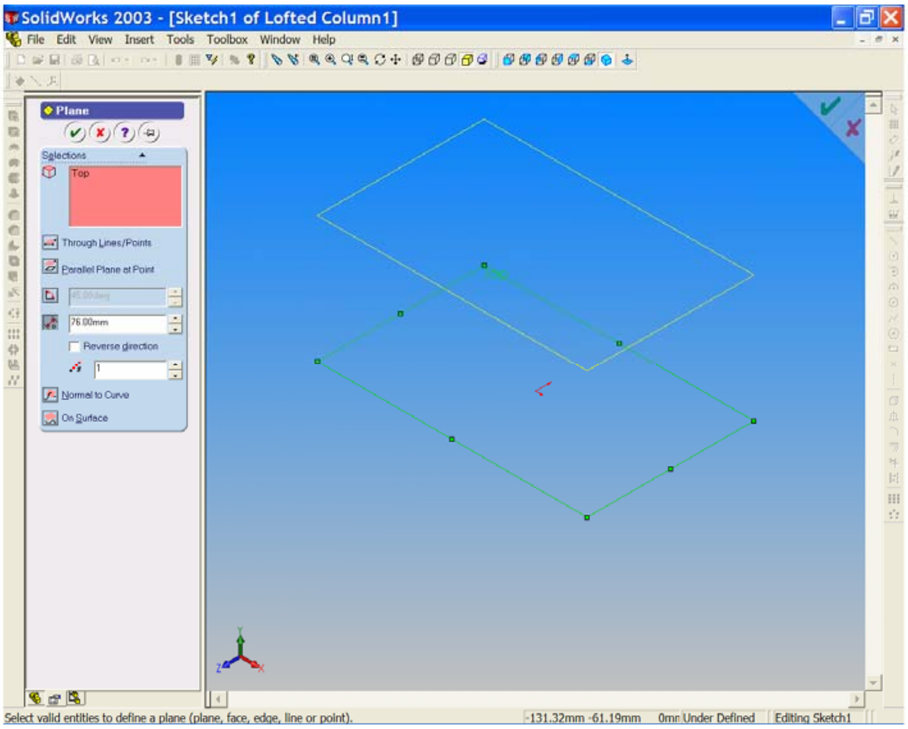


Figure 7

16. Hold down the control key and create a new plane parallel to the Top Plane and at a distance of 76mm above it. See Figure 7.

17. Open a sketch on the Top Plane and sketch a circle of Ø42 centred at the origin.

18. Open a sketch on the upper plane (plane 1) and sketch a circle of Ø32 centred at the origin. See figure 8.

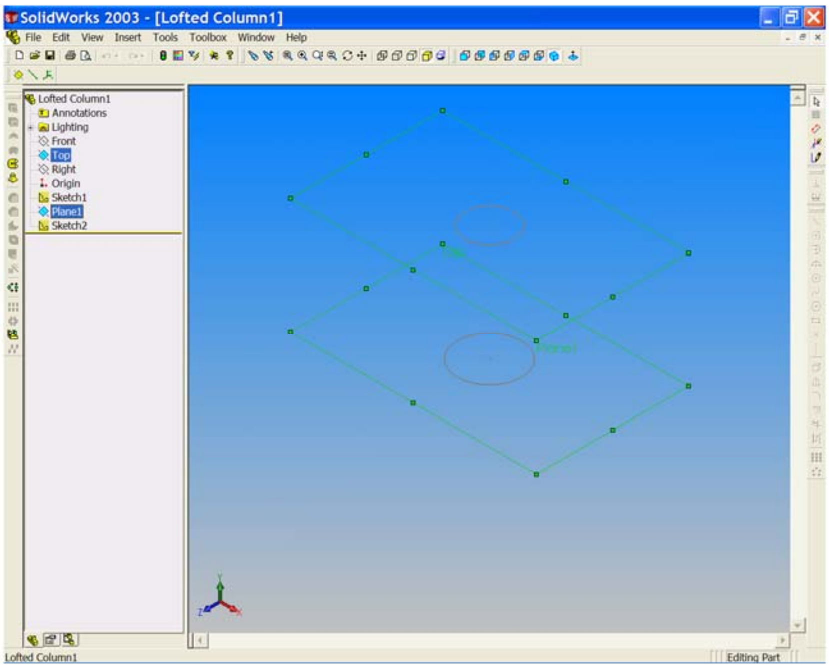


Figure 8

19. Select the ‘**Loft**’  button from the ‘**Features Toolbar**’. Ensure that ‘**Show Preview**’ is checked under options in the ‘**Loft Properties Manager**’. Select the lower circle at any point on its circumference. The point is highlighted on the sketch and is listed in the pink profiles field of the ‘**Loft Properties Manager**’.

20. Select the upper circle circumference at approximately the same position as the lower one.

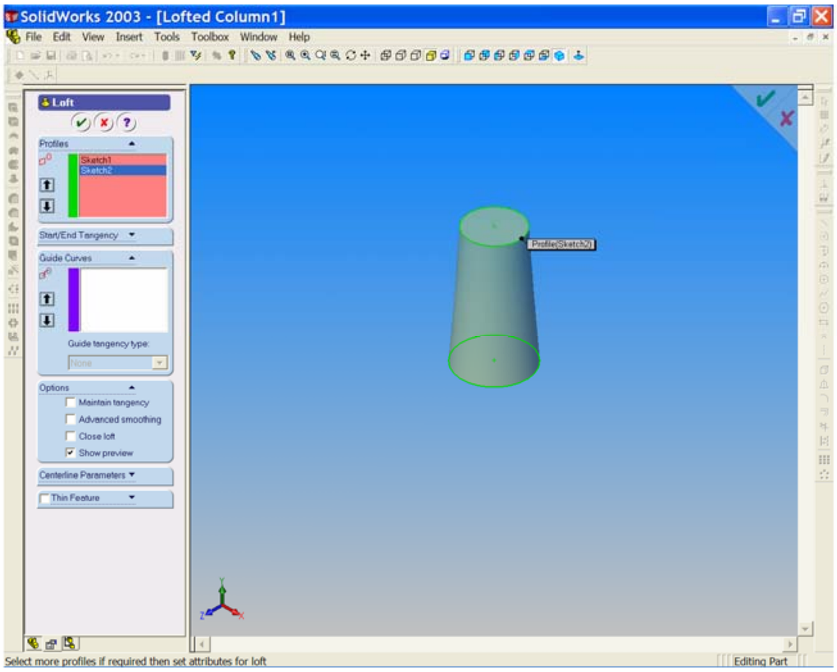


Figure 9

A solid loft is generated between the two sketches and the second sketch is listed in the pink profiles field.

21. Select ‘**OK**’  to complete the command.

22. SAVE YOUR WORK!!!

23. We can now weld all the parts together. Open the ‘**Hand Rail Column Top**’, the ‘**Flange**’ and the ‘**Lofted Column**’.

24. Choose ‘**New**’  and select ‘**Assy**’ . 

25. From the ‘**Window**’ drop down menu choose ‘**Tile Horizontally**’. See Figure 10.

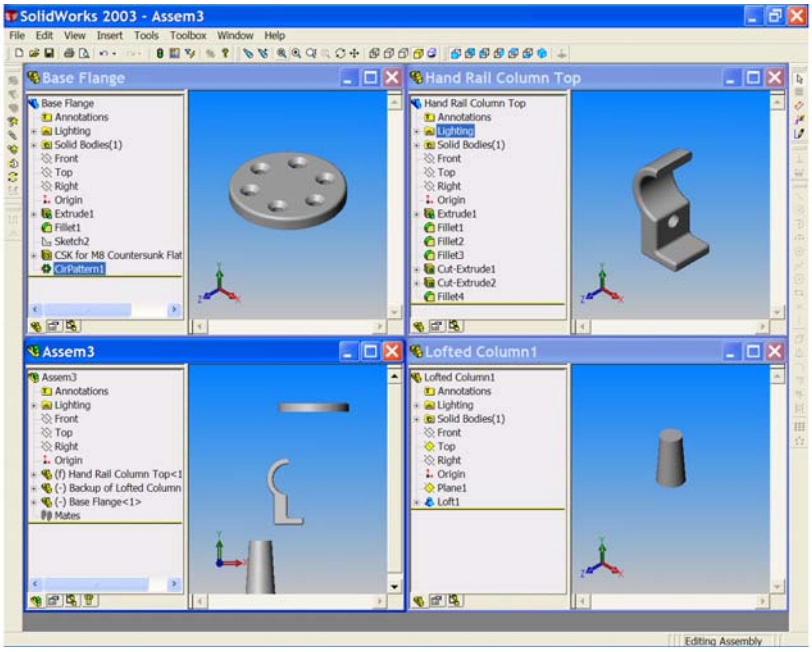


Figure 10

26. *We can now begin the procedure for joining these parts together by welding.*

    Place the '**Hand Rail Column Top**' into the Assembly first. This first part will be the '**Anchored Part**' of the assembly. From the top of the design tree drag the '**Hand Rail Column Top**' icon into the design tree area of the assembly window. Remember to hold the left button down as you drag and drop. Drag the other two parts from the top of their design trees and drop them in the graphics area of the assembly window.

27. Maximise the assembly window. To see the parts more clearly choose '**Isometric**' . See figure 11.

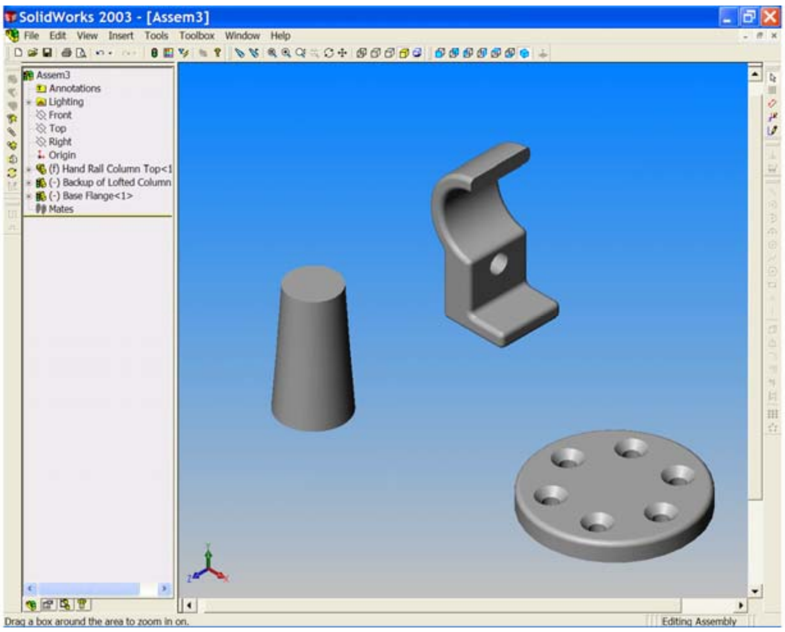


Figure 11

28. We will now mate the base of the lofted feature to the top surface of the flange. Select ‘**Mate**’ from the ‘**Assembly Toolbar**’. Choose the Top Surface of the flange to activate it and to list it in the ‘**Entities to Mate**’ field of the ‘**Mate Properties Manager**’.

29. Use the ‘**Rotate**’  button and choose the bottom of the lofted feature. Select ‘**Coincident**’ and ensure that ‘**Closest**’ is checked.

30. *We will now make these two cylindrical parts concentric.*

    Select the circular base of the Flange. Hold down the control key and select the circular base of the lofted feature. Choose ‘**Mate**’ once again; note that the two surfaces are listed and that the concentric mate is now available. Mate these two features to each other.

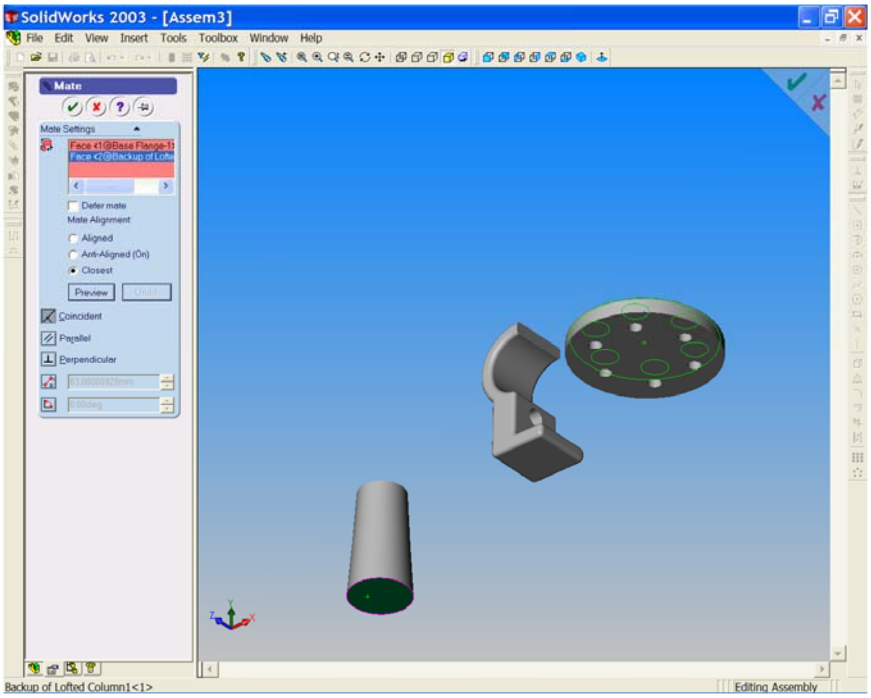


Figure 12

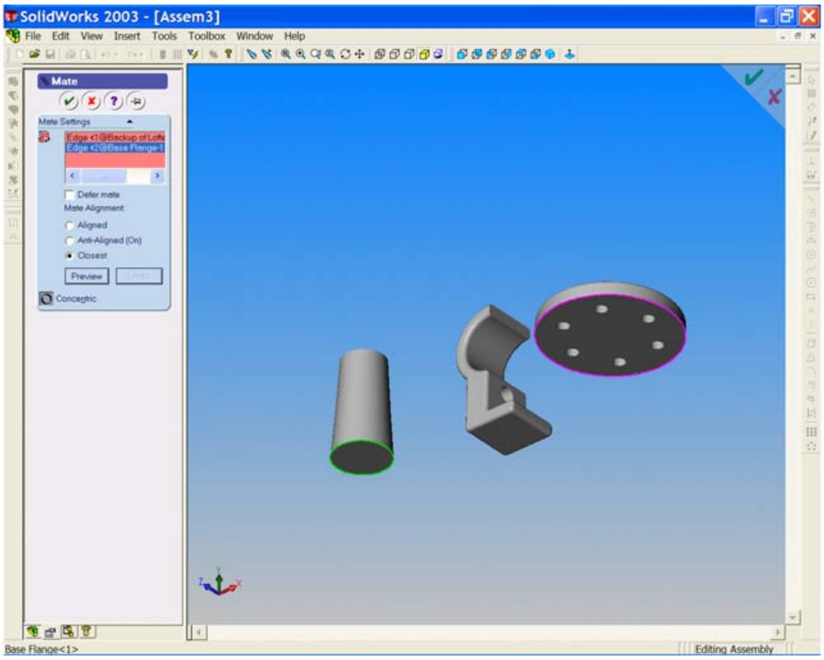


Figure 13

31. *We must now mate these two parts to the bottom surface of the final part.*

Use '**Mate**' '**Coincident**' to align the top surface of the lofted feature with the bottom surface of the final part. See Figure 14.

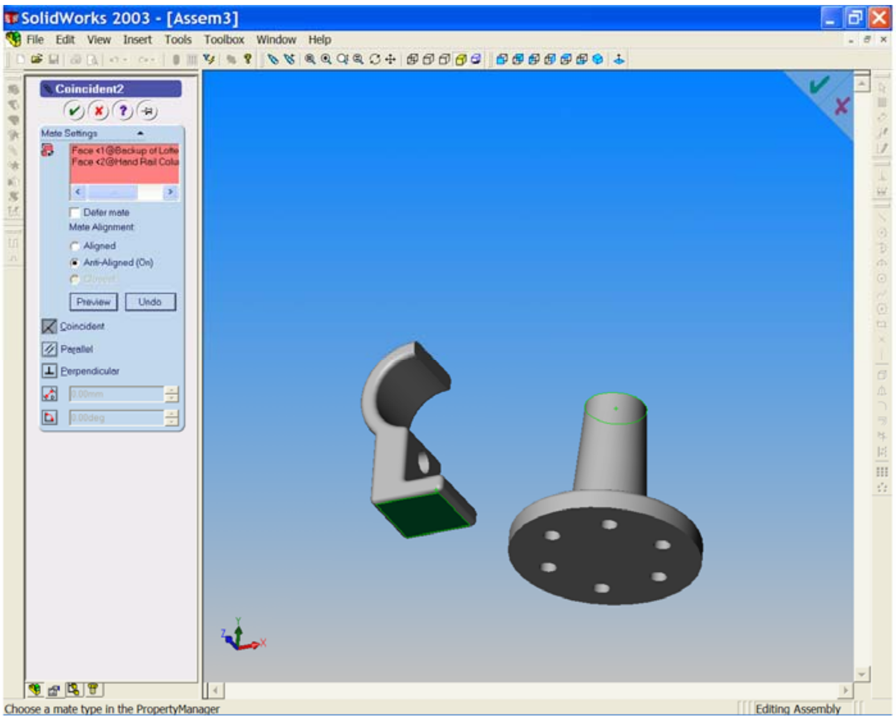


Figure 14

32. In the design tree select the '**Origin**'. From the '**View**' drop down menu, switch on '**Temporary Axes**'.

33. Select '**Mates**' note that the origin is already listed in the '**Entities to Mate**' field. Select the vertical axis through the lofted feature and choose '**Coincident**' and **Preview**' in the '**Mate Property Manager**'. See Figure 15.

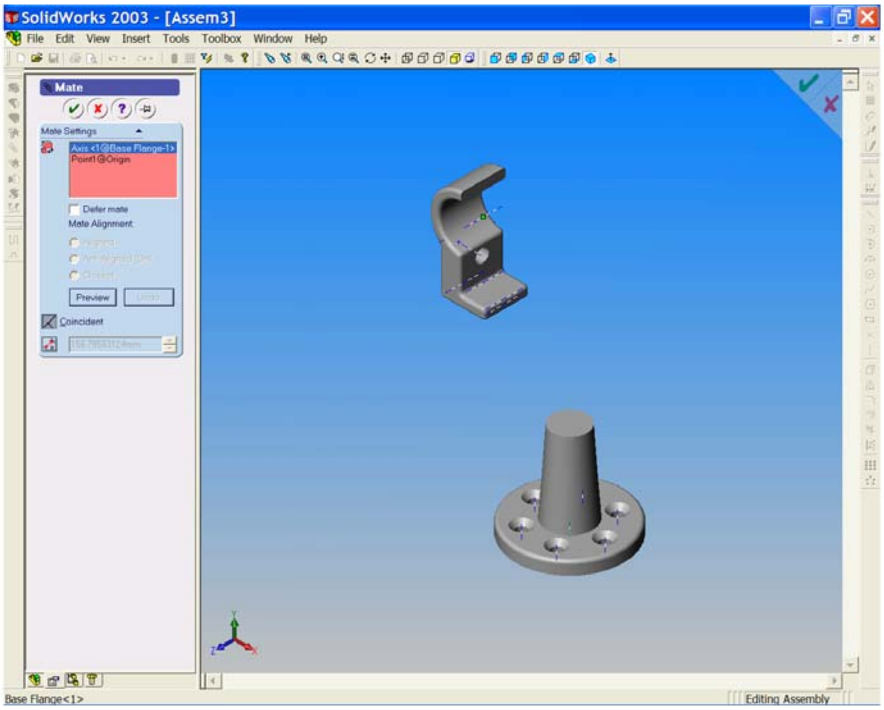


Figure 15

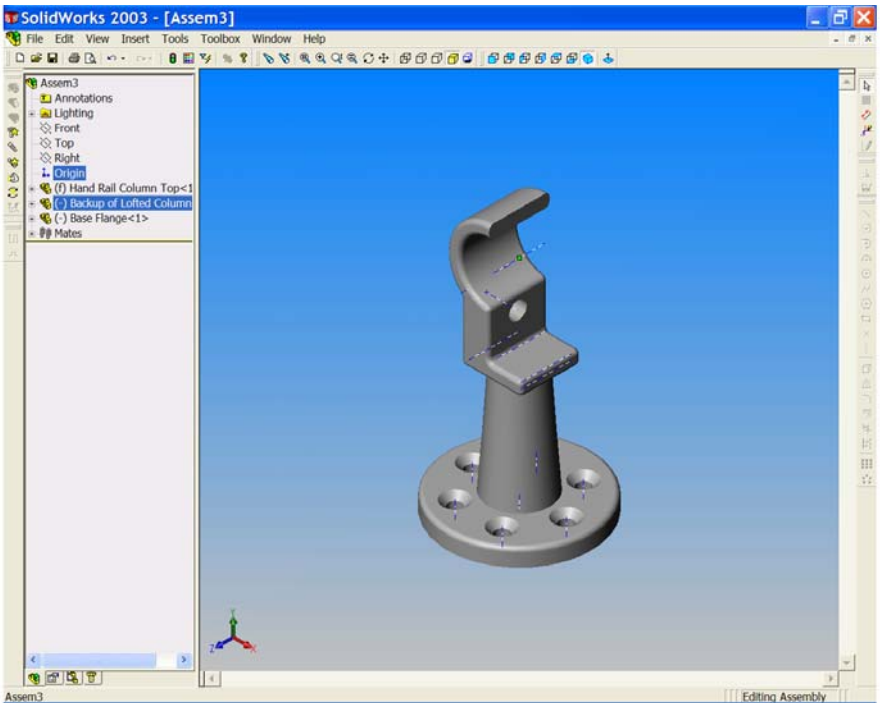


Figure 16

Notice that all parts are now correctly positioned and ready for welding. Figures 15 & 16.

Switch off the temporary axes.

SAVE YOUR WORK!!!

34. From the ‘**Insert**’ drop down menu navigate to ‘**Weld Bead**’. See Figure??? The weld bead window opens. The most suitable weld here is the ‘**Fillet Weld**’. Choose ‘Fillet’ from the ‘ISO’ ‘Weld Types’ offered in the window and choose ‘**Next**’.

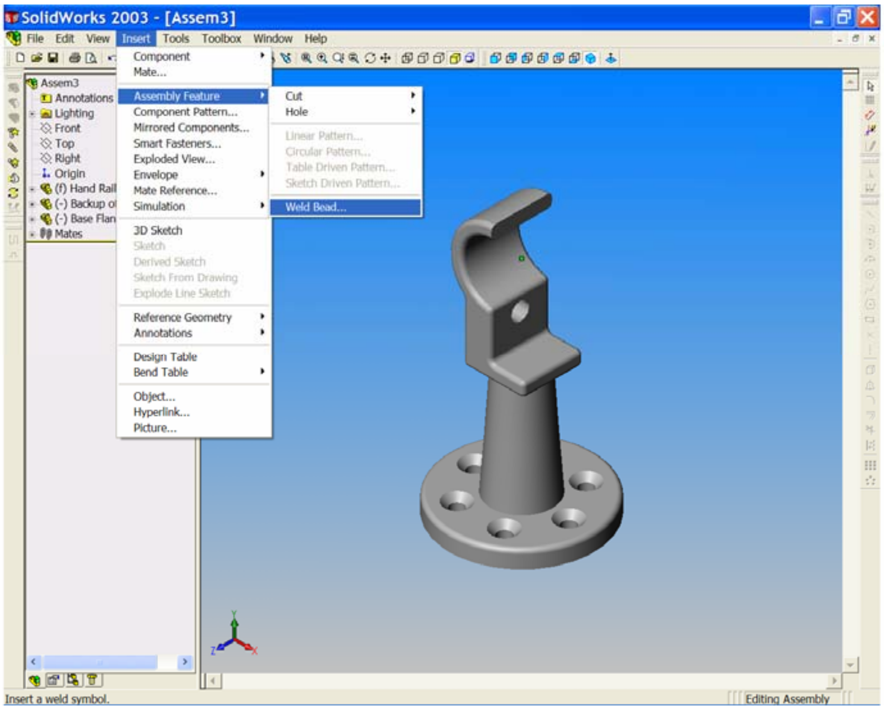


Figure 17

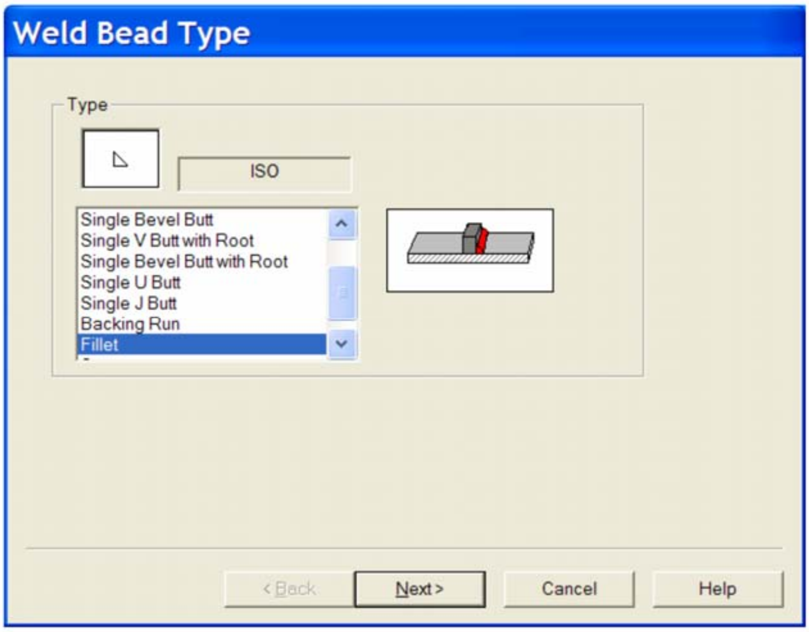


Figure 18

(Note the weld symbol and the pictorial diagram change as you make selections.)

- For ‘**Surface Shape**’ choose ‘**Concave**’.
- For ‘**Top Surface Delta**’ increase the value to **0.75**.
- For ‘**Radius**’ increase the value to **5mm**.

When the above changes are in place choose ‘**Next**’.

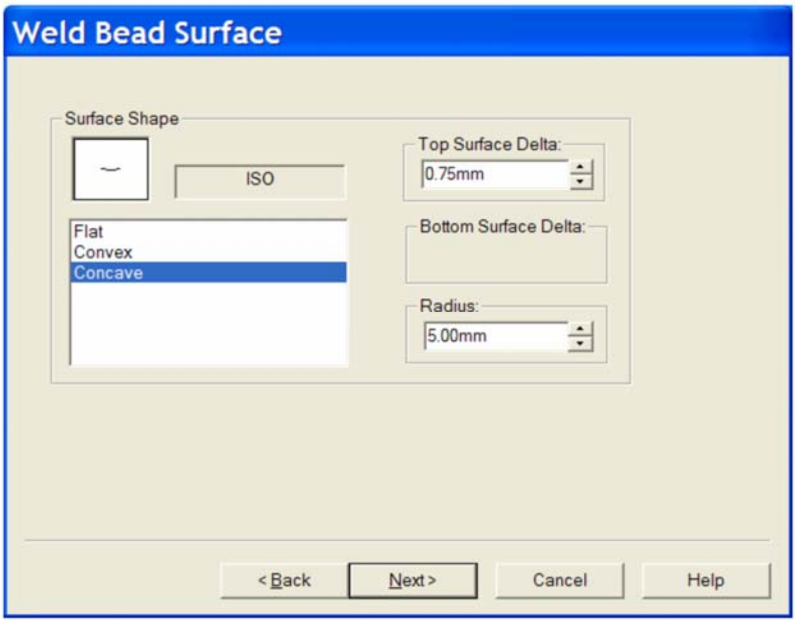


Figure 19

We must now select the surfaces to be welded.

Select the top surface of the Flange and the cylindrical surface of the Lofted Column. These two surfaces are listed in the pink ‘**Contact Faces**’ window. Choose ‘**Next**’ and ‘**Finish**’. The weld is generated between the two parts and is added to the design tree. See Figure 20.

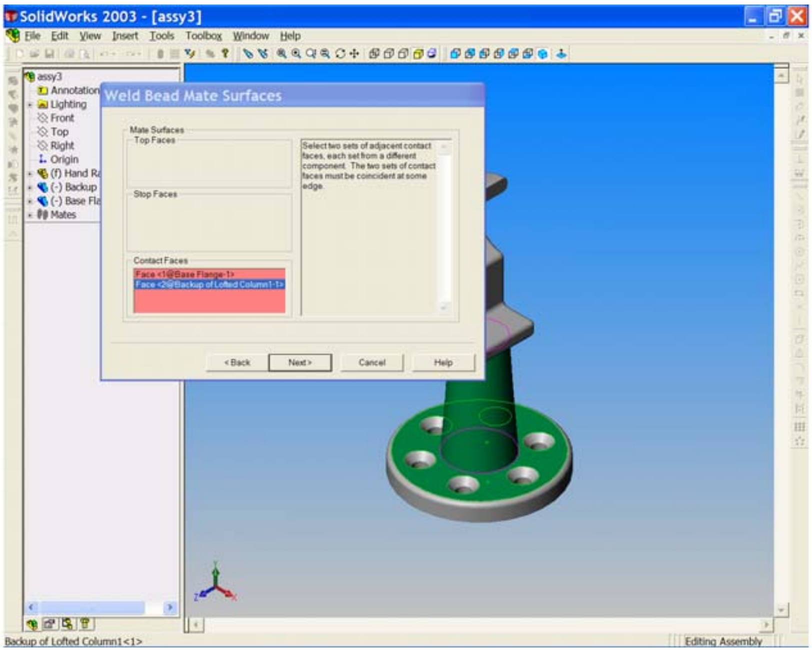


Figure 20

35. With the ‘**Rotate**’ tool manoeuvre the model so that the under surface of the model is visible and apply the same weld between the Top of the lofted column and the bottom of the other piece.

You will find however that the 5mm radius is too big for a weld at this location.

Use the back button to return to the appropriate window and reduce the radius to 3mm. The weld will now be satisfactory. See Figure 21.

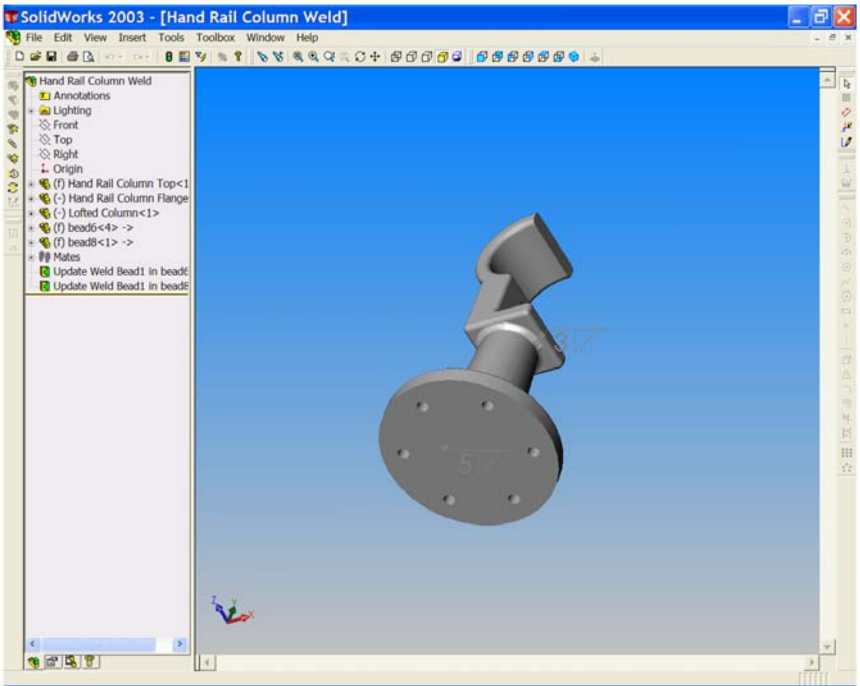


Figure 21

SAVE YOUR WORK!!

Module 15: Inserting Standard Parts from the ‘Toolbox’

1. Open the ‘**Hand Rail Column**’ assembly file.

2. Open the ‘**Hand Rail Column Cap**’ part file.

3. Drag & drop the ‘**Cap**’ into the ‘**Assembly**’ file.

4. Use ‘**Coincident**’ and ‘**Concentric**’ Mates to align the Cap with the Main Part.

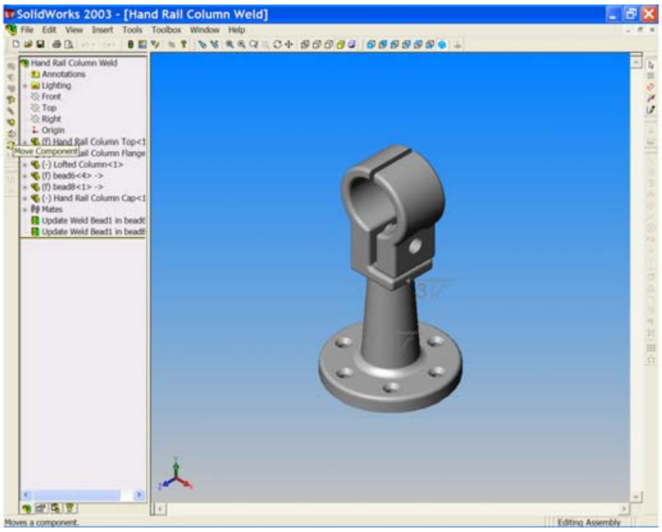


Figure 1

5. Use ‘**Move Component**’ to position the Cap as shown in Figure 1.

*The parts are now aligned so that a standard **Bolt, Nut and Lock Nut** may be added. We may gain access to the ‘**Toolbox Browser**’ by selecting the tab at the bottom of the ‘**Feature Manager Design tree**’. See Figure 2.*

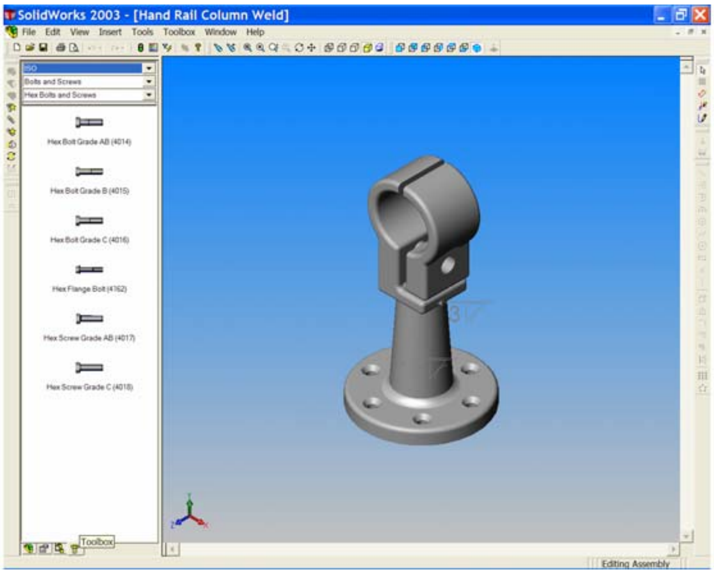


Figure 2

6. When the ‘**Toolbox Browser**’ is open we may drag and drop many ‘**Standard Parts**’ into an assembly. In this exercise we are going to use a standard **M12 x 65 lg. Bolt, Nut and Lock Nut**. At the top of the Toolbox Browser there are three drop down windows that are used to navigate to the parts we require. To drag and drop these parts we must put some settings in place;

    - Choose ‘**ISO**’ from the ‘**Catalog**’ window (top window).
    - Choose ‘**Bolts and Screws**’ from the ‘**Chapter**’ window (centre window).
    - Choose ‘**Hex Bolts and Screws**’ from the ‘**Page**’ window (bottom window).

7. With the ‘**Rotate**’  button, manoeuvre the parts so that you can see the bolt hole in the main feature. See Figure 3.

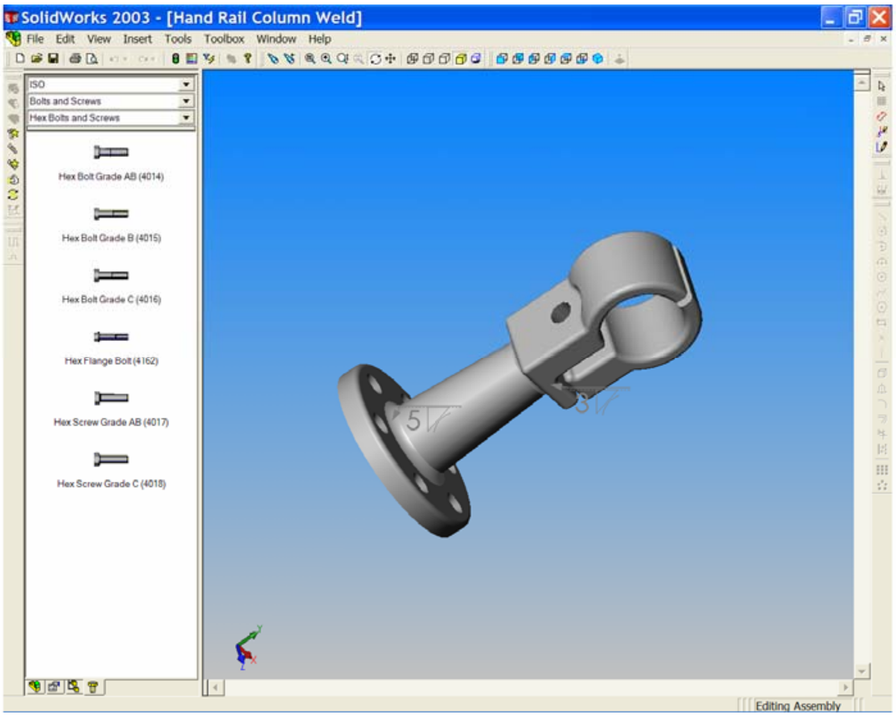


Figure 3

8. Drag a ‘**Hex bolt Grade AB (4014)**’ from the Toolbox Browser to the graphics area and drop it into the bolt hole of the main part.

   *The software will apply a ‘**Smart Mate**’ between the hole and the bolt and you will see the bolt snap into place.*

9. The bolt dialog box appears. Set the parameters of the bolt as follows;

   - Size ‘**M12**’
   - Length ‘**65**’
   - Thread Length ‘**30**’
   - Thread Display ‘**Schematic**’

   Left click OK to add the bolt to the assembly. See Figures 4 & 5.

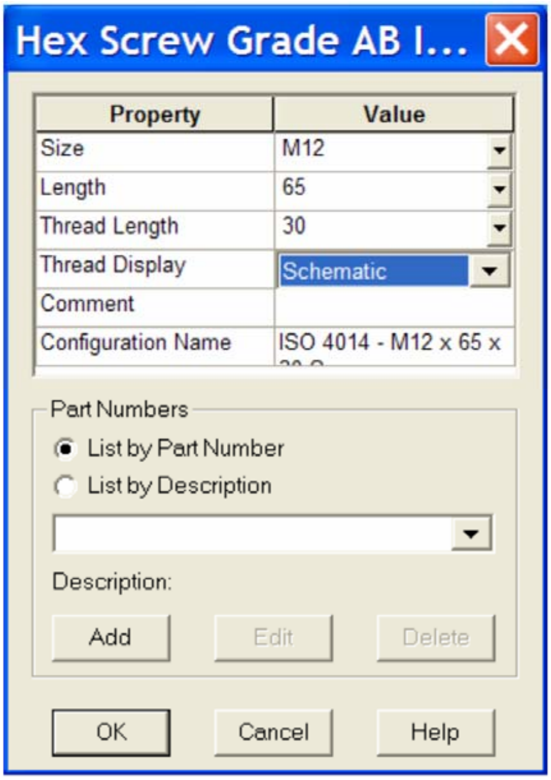


Figure 4

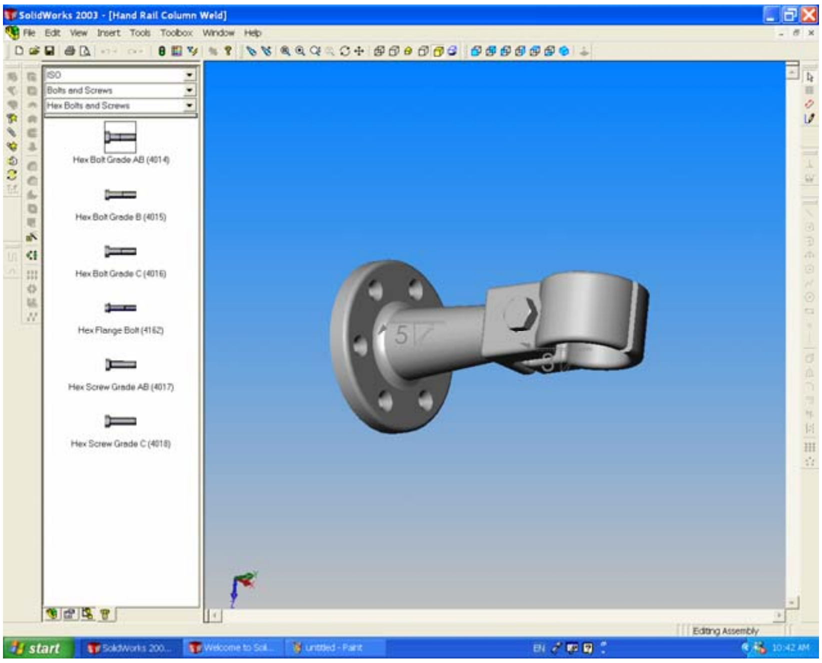


Figure 5

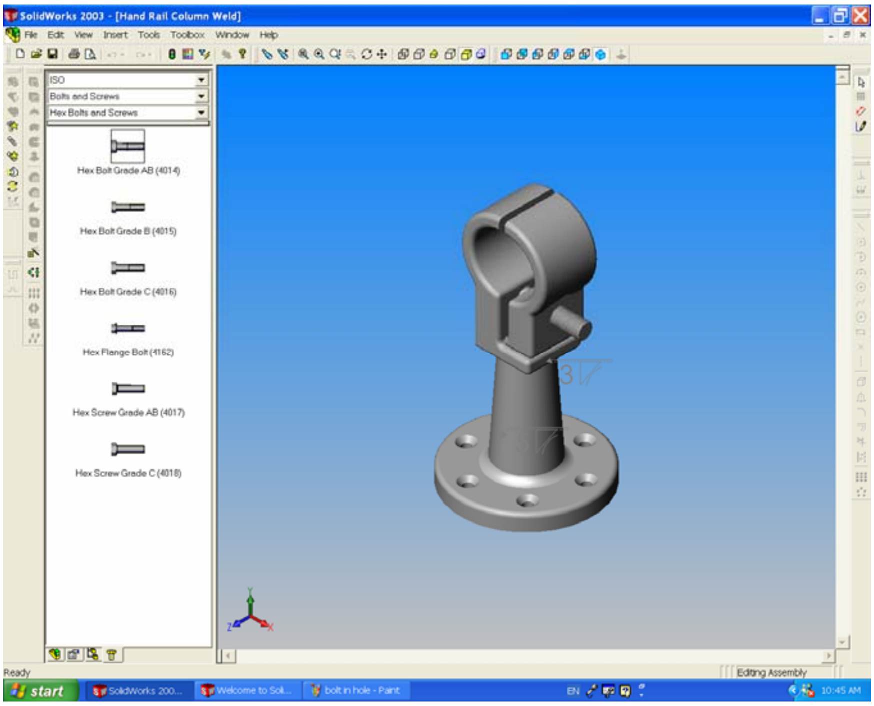


Figure 6

10. Select ‘**Isometric**’. You can now see the Bolt passing through the two parts. See Figure 6. From the ‘**Chapter**’ window choose ‘**Nuts**’. The ‘**Page**’ is automatically selected and opens at ‘**Hex Nuts**’. Drag a ‘**Grade C (4034) Nut** into the assembly and drop it on the edge of the hole where the bolt protrudes. Set the parameters of the nut to suit those of the bolt;

- Size ‘**M12**’
- Thread Display ‘**Schematic**’

Left click OK to add the nut. See Figures 7 & 8. **SAVE YOUR WORK!!!**

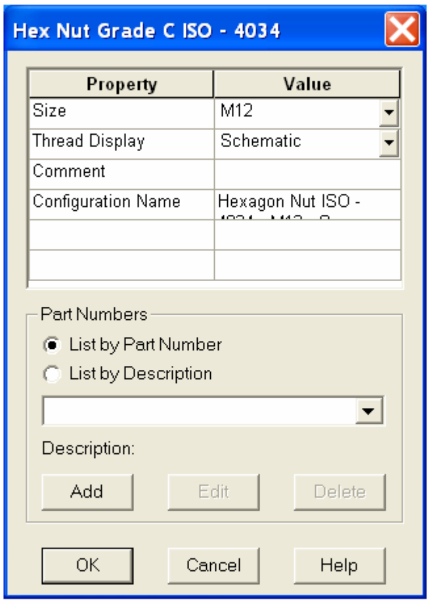


Figure 7

11. Drag a ‘Thin (Chamfered) (4035)’ Nut and drop in the graphics area of the screen. Once more enter the nut parameters to match the standard nut. Click OK to place the nut near to the assembly. See Figure 8.

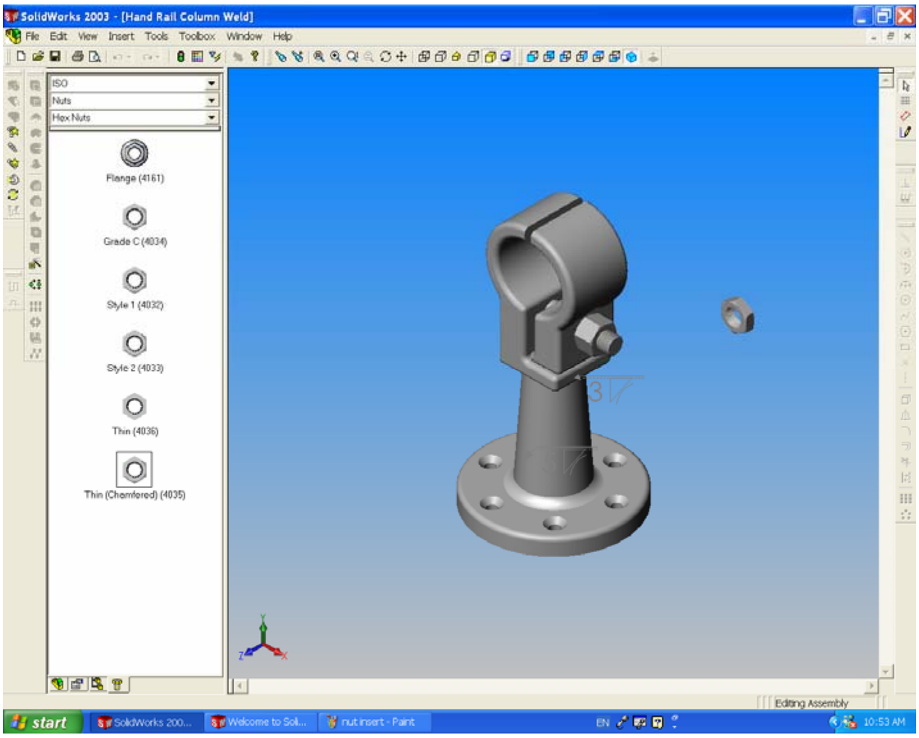


Figure 8

12. Mate the locknut coincident to the end face of the Standard Nut and concentric with the bolt. See Figures 9, 10 & 11.

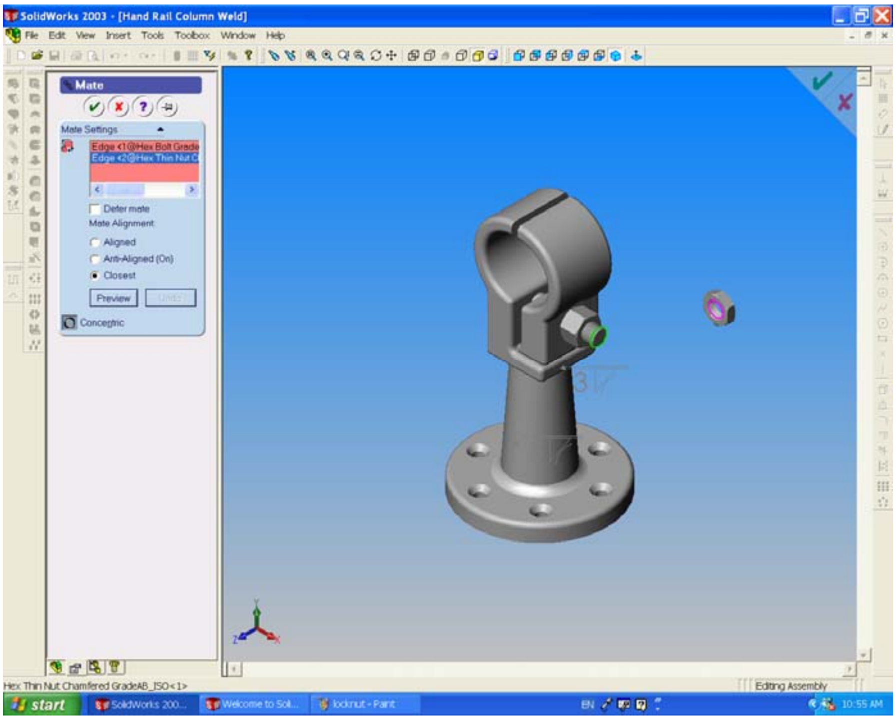


Figure 9

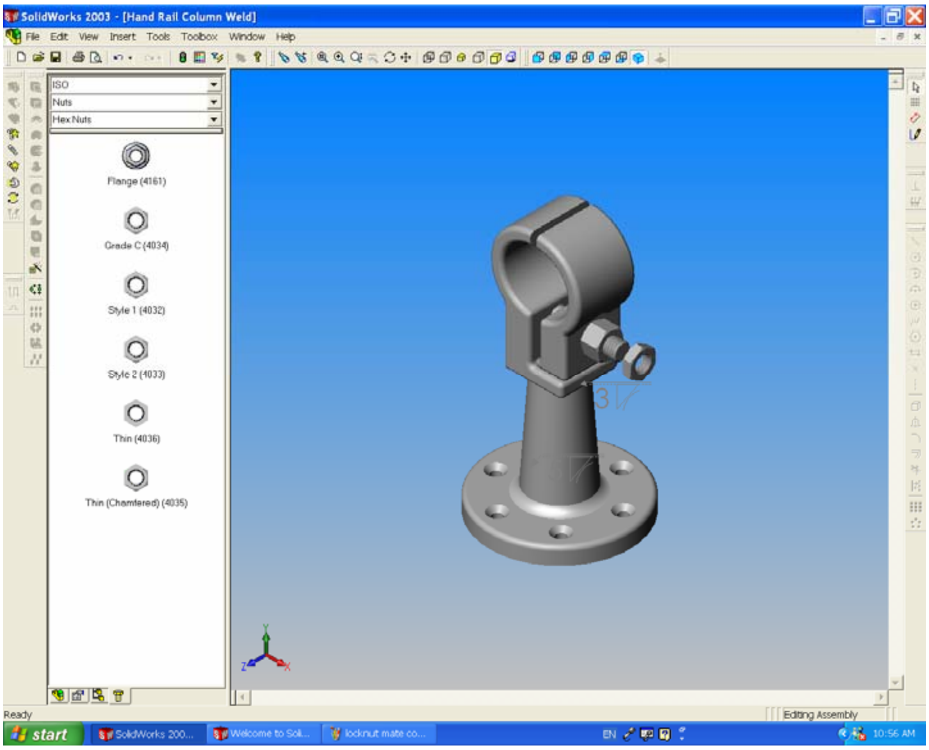


Figure 10

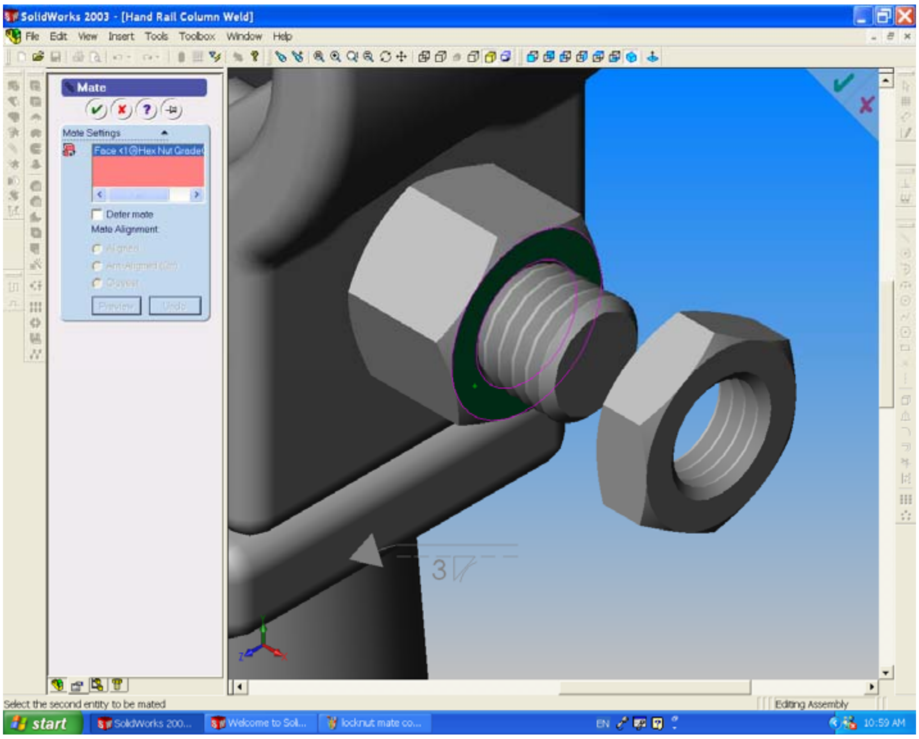


Figure 11

13. The assembly is now complete. See Figure 12. **SAVE YOUR WORK!!**

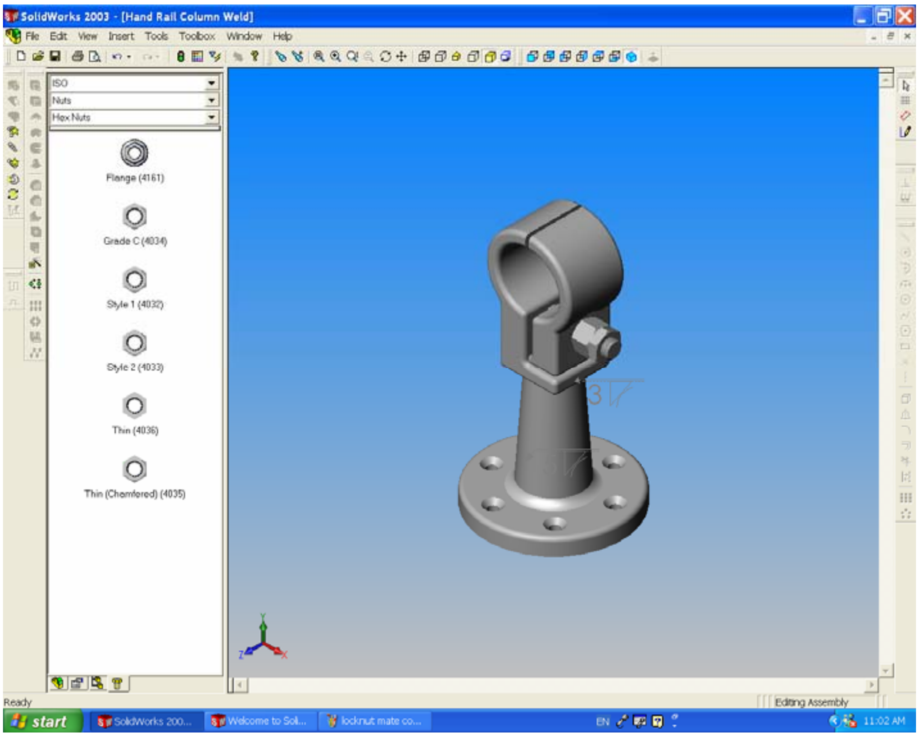


Figure 12

Module 12: Sweep Features I; Cranking Lever

The cranking lever details are shown in Figure 1.

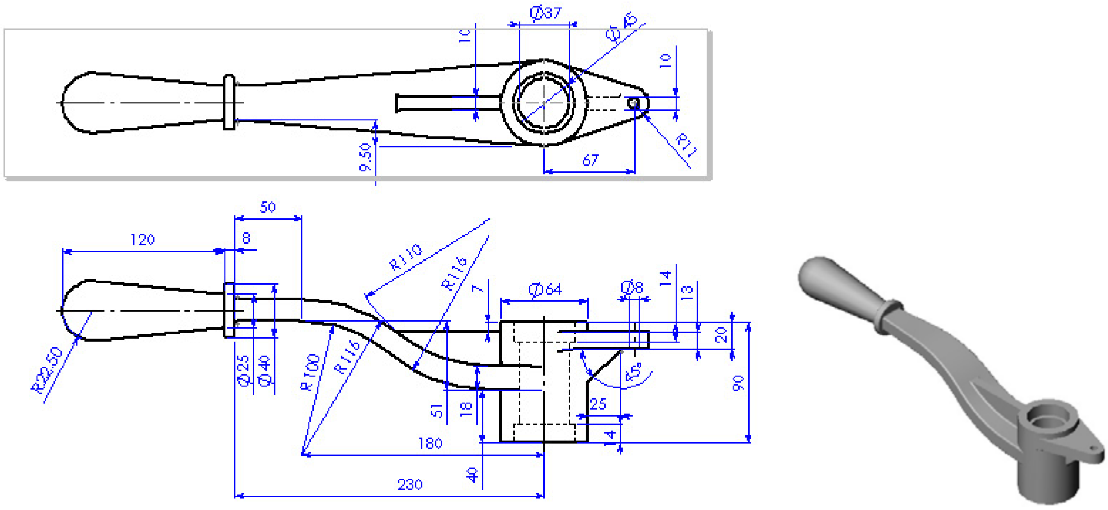


Figure 1 Cranking lever

It can be modeled by the following steps as shown in Figure 2.

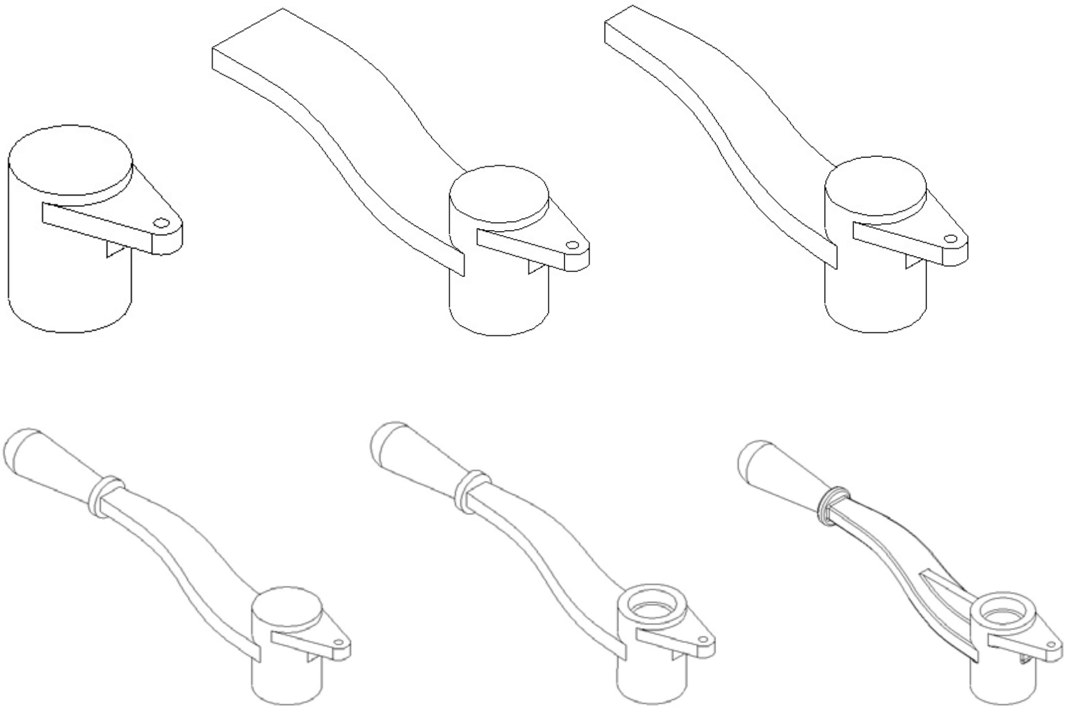


Figure 2 Cranking lever modeling process.

1. Create a cranking lever head using a revolved boss/base feature.
2. Create a crank lever head with an extruded boss/base and revolved cut
3. Add a handle to the handle body using the revolved boss/base.
4. Create two ribs fillets on the crank lever.

Create a Cranking Lever Head

1. Start SolidWorks.
2. Select **New** on the Standard toolbar, or click **File, New** on the menu bar.
3. Select **Part** from the **Template** tab in the dialog box, and click **OK.**
4. Click **File, Properties** on the menu bar.
5. Fill in the necessary properties in the dialog box, and click **OK** to accept the properties and close the dialog box.
6. Click **Tools, Options** on the menu bar to open the option dialog box.
7. Select the **Document Properties** tab.
8. Click **Unit** in the properties tree text box.
9. Select **Millimeter** and set the decimal places to 2.
10. Click **OK** to close the Properties dialog box.
11. Select **Sketch** on the Sketch toolbar to open a new sketch on the **Front** Plane.
12. Select **Centerline** on the Sketch Tools toolbar to draw a vertical centerline through the origin as the axis of the revolution.
13. Select **Line** on Sketch Tools toolbar, and draw and dimension a profile of the cranking lever head as shown in Figure 3.

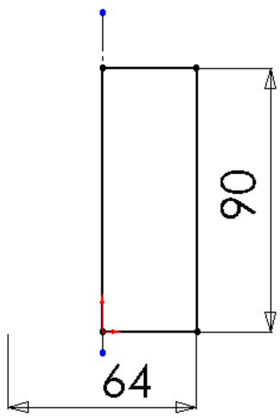


Figure 3 Cranking Lever Head Sketch

14. Select **Revolved Boss/Base** on the Features toolbar, and click **OK** .
15. Select the **Front** plane on the FeatureManager design tree.
16. Select **Sketch** on the Sketch toolbar to open a new sketch for the lever head rib.
17. Click **Normal To** on the Standard Views toolbar to make the sketch plane parallel to the screen.
18. Sketch and dimension the lever head rib as shown in Figure 3.

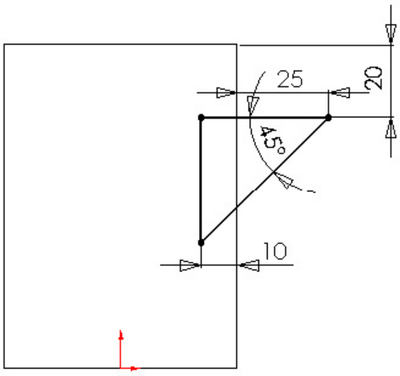


Figure 3 Lever head rib sketch.

19. Select **Extruded Boss/Base** on the Features toolbar.
20. Select **Mid Plane** on the **End Condition** drop down list box.
21. Type **10mm** in the depth box under **Direction 1**, and click **OK**.
22. Select the top surface of the rib as shown in Figure 4, and open a sketch.

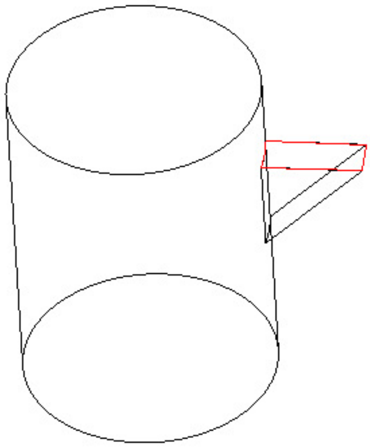


Figure 4 Sketch plane for lever head ear.

23. Sketch and dimension the lever head ear as shown in Figure 5.

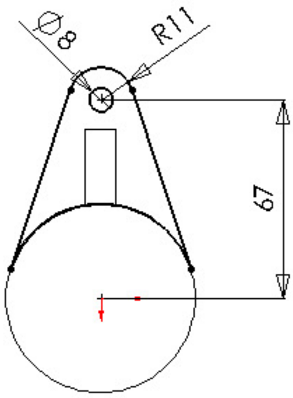


Figure 5 Lever Head Ear Profile

24. Select **Extruded Boss/Base**  on the Features toolbar.

25. Enter **13 mm** in the depth box under **Direction 1** as in Figure 6.

26. Click **OK** .

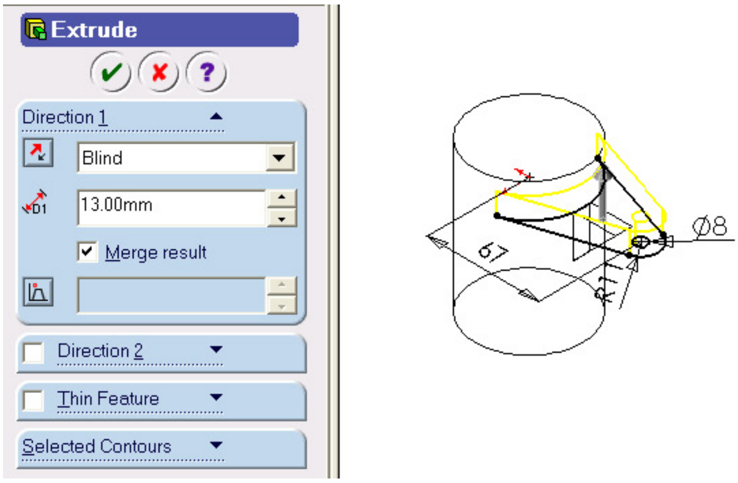


Figure 6 Lever Head Ear Extrusion

Create a Handle Body using Sweep

1. Select the **Front** on the FeatureManager design tree.

2. Select **Sketch**  on the Sketch toolbar to open a new sketch for the sweep guide curve.

3. Sketch and dimension the sweep guide curve as shown in Figure 7.

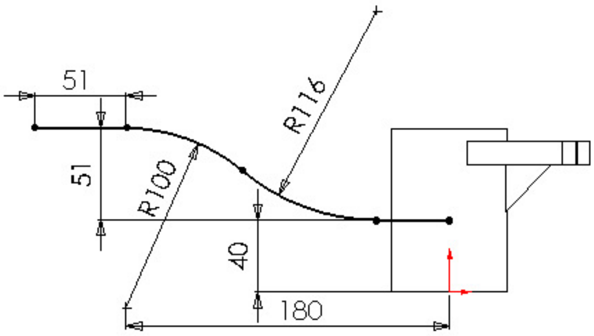


Figure 7 Sweep path

4. Click **Confirmation Corner**  or select **Rebuild**  on the Standard toolbar to exit the sketch.

5. Select **Front** on the FeatureManager design tree.

6. Select **Sketch**  on the Sketch toolbar to open a new sketch for the sweep guide curve.

7. Sketch and dimension the sweep guide curve as shown in Figure 8.

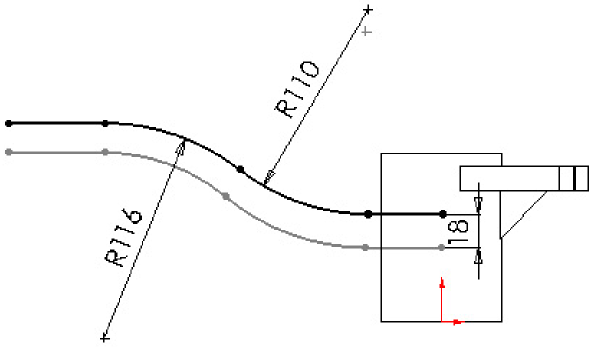


Figure 8 Sweep guide curve.

8. Click **Confirmation Corner**  or select **Rebuild**  on the Standard toolbar to exit the sketch.

9. Select **Insert, Reference Geometry**, **Plane** on the menu bar or Click **Plane**  on Reference Geometry Toolbar.

10. Click the **Offset Distance**  button.
11. Select the **Right** plane on the FeatureManager design tree, and enter **230 mm** in the Distance  box.
12. Check the **Reverse direction** check box as shown in Figure 9.

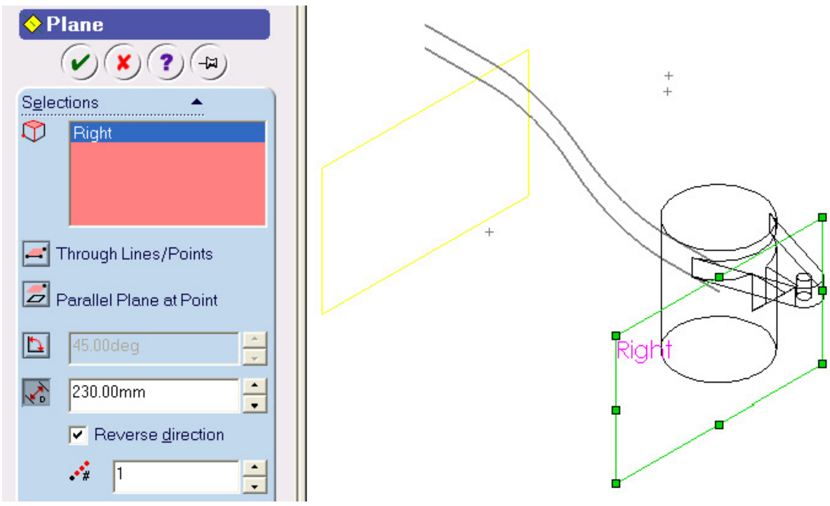


Figure 9 Reference plane for sweep profile

13. Click **OK** .
14. Select the **Plane1** on the FeatureManager design tree or in the graphic window.
15. Select **Sketch**  on the Sketch toolbar to open a new sketch for the sweep profile.
16. Sketch a rectangle. Then, sketch 2 points on the middle of the horizontal edges.
17. Click **Add Relation**  on the Sketch Relation toolbar.
18. Apply **Midpoint**  constraint to these two points.

19. Select the sweep path, and the bottom point, and apply **Pierce** constraint as shown in Figure 10.

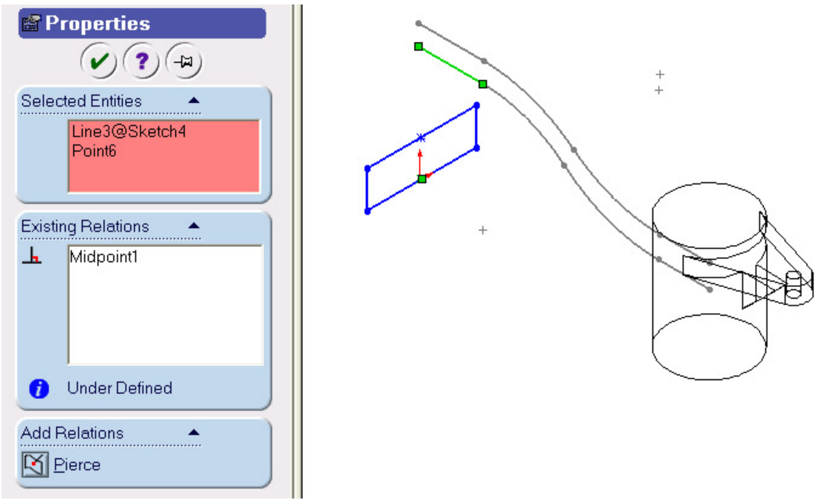


Figure 10 Pierce Constraints between Profile and the Path

20. Select the guide curve, and the top point, and apply **Pierce** constraint as shown in Figure 11.

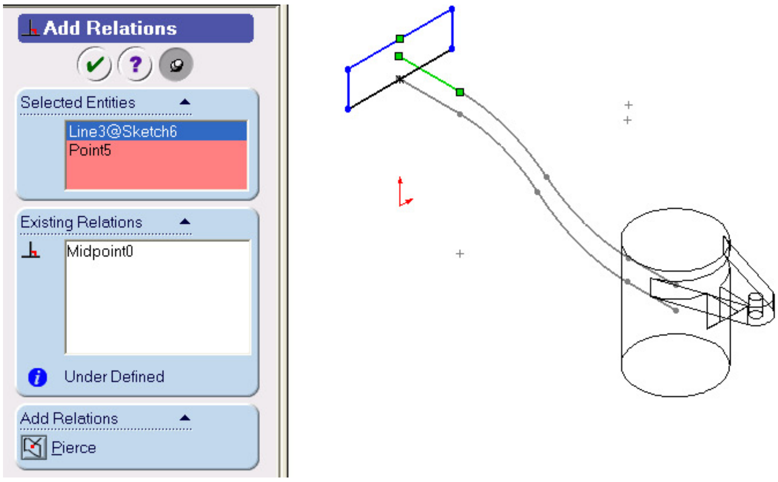


Figure 11 Pierce Constraint between Profile and Guide Curve

21. Dimension the width of the rectangle to **64 mm** as shown in Figure 12.

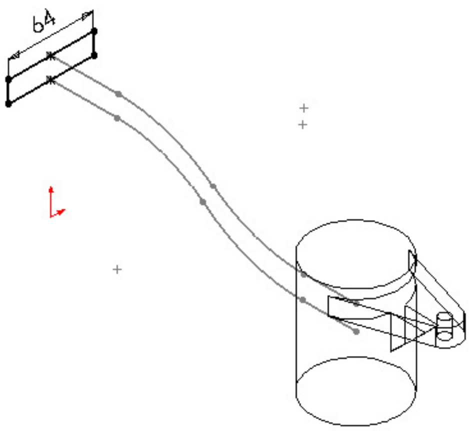


Figure 12 Profile dimension

22. Click **Confirmation Corner** or click **Rebuild** on the Standard toolbar to exit the sketch.

23. Select **Sweep Boss/Base** on the Features toolbar.

24. Select the path **(Sketch 4)** in the path box under the Profile and Path frame.

25. Select the guide curve in the **Guide Curves** 4 box as in Figure 13.

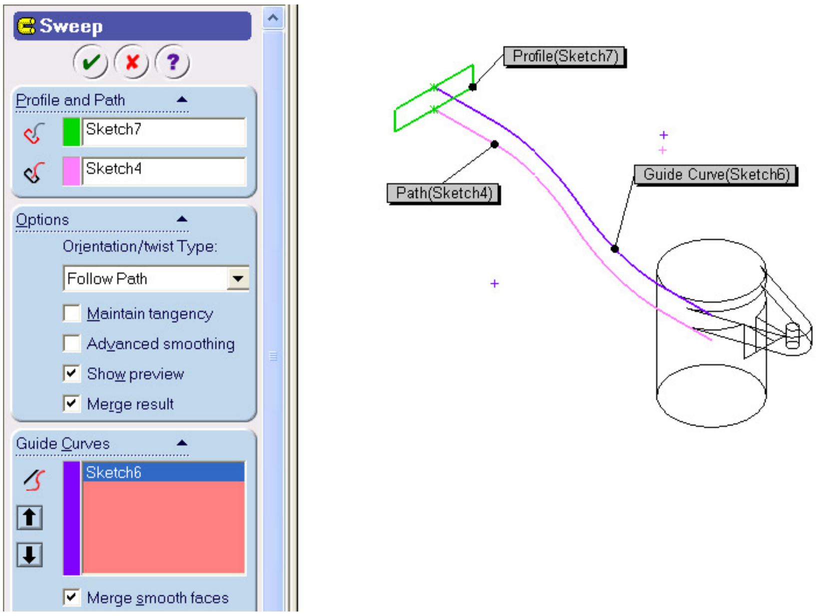


Figure 13 Sweep parameters.

26. Select **OK**.

27. Select the bottom surface of the lever head as shown in Figure 14, and open a sketch.

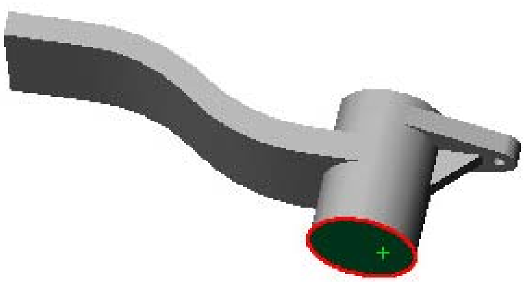


Figure 14 Sketch plane for lever handle cut.

28. Select the head circle, and click **Convert Entities** on the Sketch Tools toolbar to create a circle on the sketch plane.

29. Sketch three lines, and **Trim** the circle to form the cut profile as shown in Figure 15.

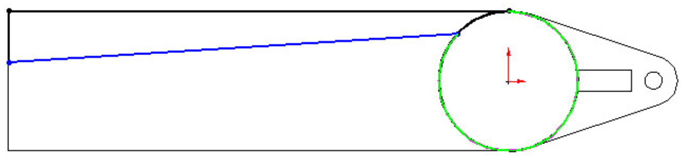


Figure 15 Lever Body Cut Sketch

30. Select **Add Relation**  on the Sketch Relations toolbar. Select the arc and the angled line, and apply a **Tangent**  constraint to them.
31. Dimension the sketch as shown in Figure 16.

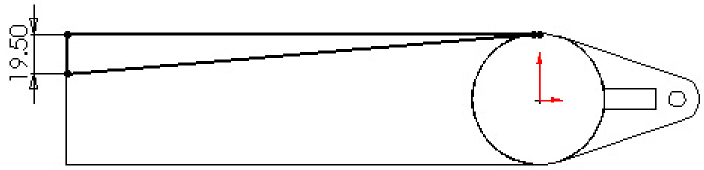


Figure 16 Lever body cut profile

32. Select **Extruded Cut**  on the Features toolbar. Select **Through All** on the end condition drop down list box, then click **OK** 
33. Select the last Feature created in FeatureManager Design Tree, i.e., **Cut-Extrude 2,** you may or may not have a different name for this Feature.
34. Select **Mirror**  on the Features toolbar. The **body cut (Cut-Extrude 2)** should be selected as the **mirror feature**. If not, select it. Then select **Front** plane on the FeatureManager design tree as the **mirror** plane.
35. Click **OK**  to mirror the cut to the other side as shown in Figure 17.

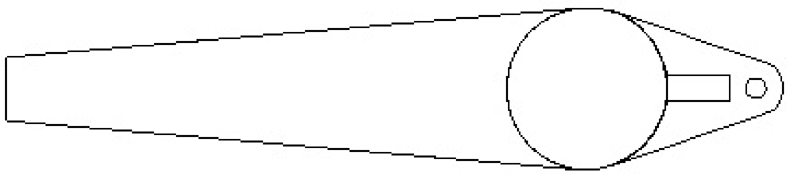


Figure 17 Lever body

Create Lever Handle and Head Hole Using Revolved Features

1. Select the **Front** plane on the FeatureManager design tree to open a sketch for the lever handle.
2. Sketch and dimension the profile as shown in Figure 18.

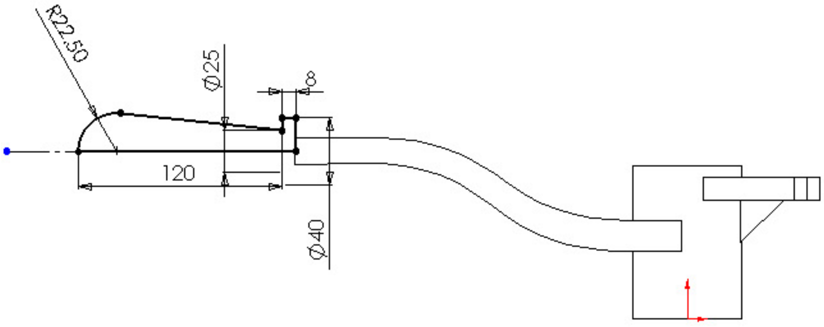


Figure 18 Lever handle profile.

3. Select **Revolved Boss/Base**  on the Features toolbar.
4. Click **OK**  to create the revolved lever handle.
5. Select the **Front** plane again in the FeatureManager design tree to open a sketch for the head hole cut.
6. Sketch the hole cut profile as shown in Figure 19.

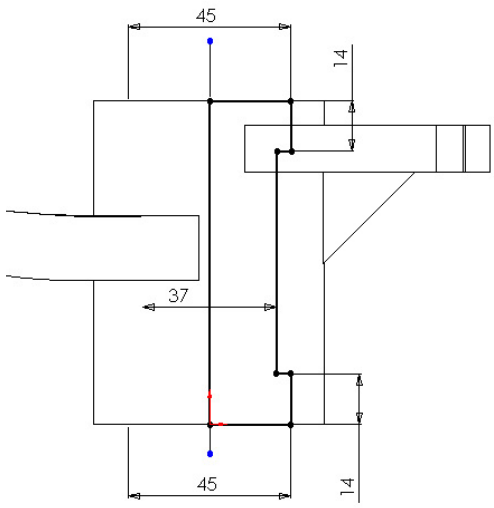


Figure 19 Head hole profile.

7. Select **Revolved Cut**  on the Features toolbar, and Click **OK**  create the head hole

Create Handle Rib and Fillets

1. Select the **Front** plane in the FeatureManager design tree to open a sketch for the rib.
2. Sketch and dimension the profile using **Line**  as shown in Figure 20.

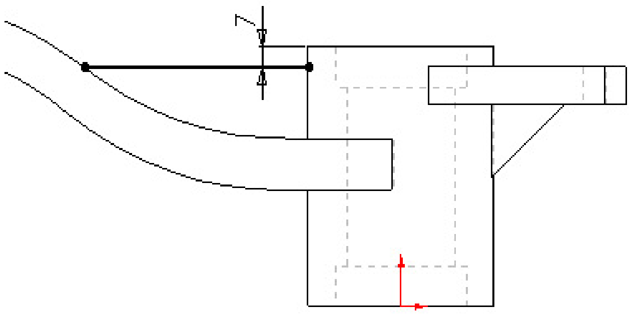


Figure 20 Handle rib profile.

3. Select **Rib** on the Features toolbar. Select **Both Sides**, enter **10 mm** in the **Rib Thickness** box and check **Flip material side** as shown in Figure 21.

4. Click **OK** to create the rib.

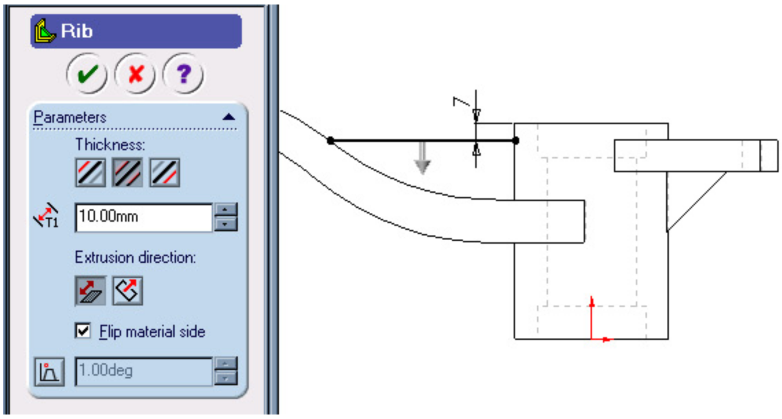


Figure 21 Handle rib parameters.

5. Select **Fillet** on the Features toolbar, and enter **2 mm** in the **Radius** box.

   Select edges around the rib edges, handle edges as in Figure 22. Then click **OK** to create fillets. The final model for Cranking Lever is shown in Figure 23.

6. **Save** the Model.

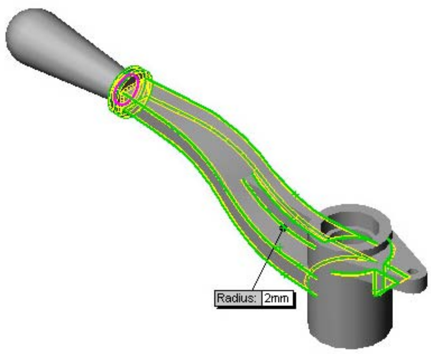


Figure 22 Cranking Lever fillets.

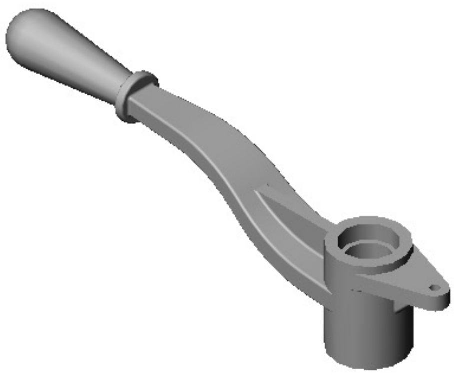


Figure 23 Cranking Lever model.